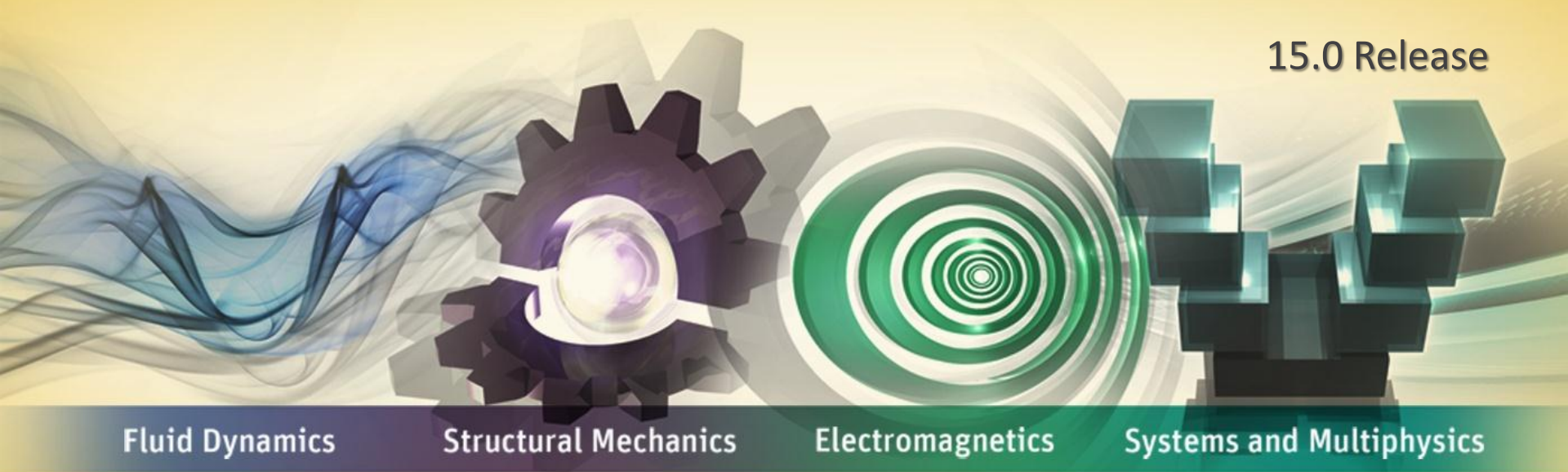


Workshop 4.2

Tetra Meshing - Valve

15.0 Release

A horizontal banner with a yellow-to-orange gradient background. It features four main sections: 'Fluid Dynamics' with a blue wavy line, 'Structural Mechanics' with a purple gear, 'Electromagnetics' with a green concentric circle, and 'Systems and Multiphysics' with a 3D bar chart. The text '15.0 Release' is in the top right.

Fluid Dynamics

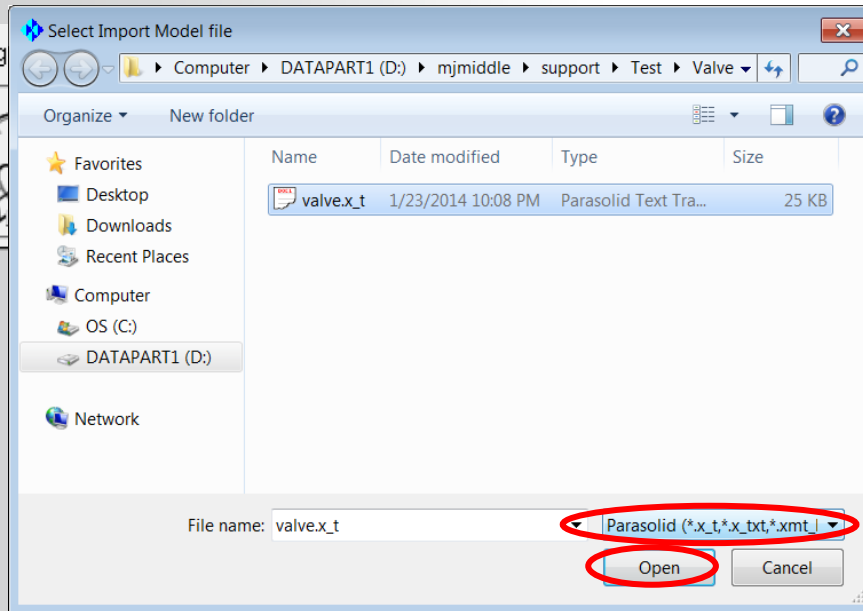
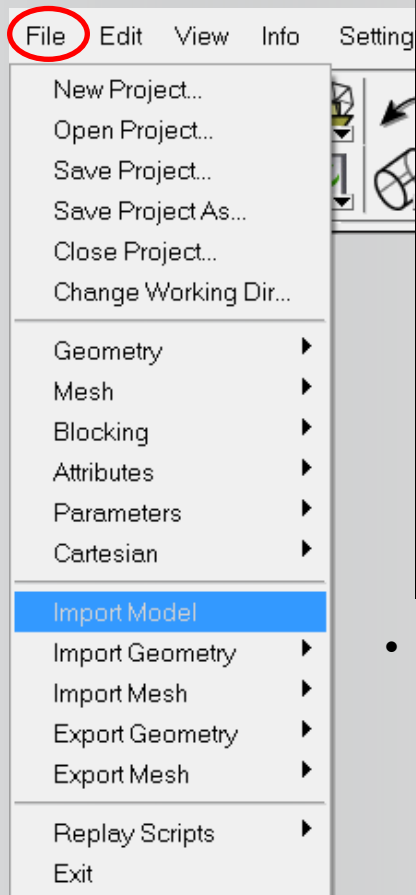
Structural Mechanics

Electromagnetics

Systems and Multiphysics

Introduction to ANSYS ICEM CFD

Import Parasolid

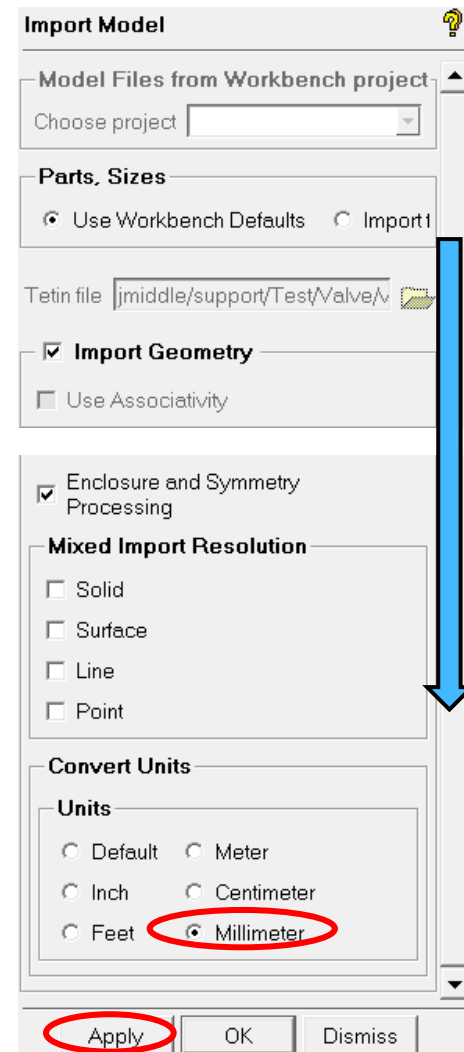


- Set working directory
 - **File > Change Working Dir...**
 - Choose **Valve**
 - **OK**

ICEM CFD has no units, but parasolid format is always in Meters, so you need it to scale the numbers back to the unit system it was when translated from CAD to parasolid

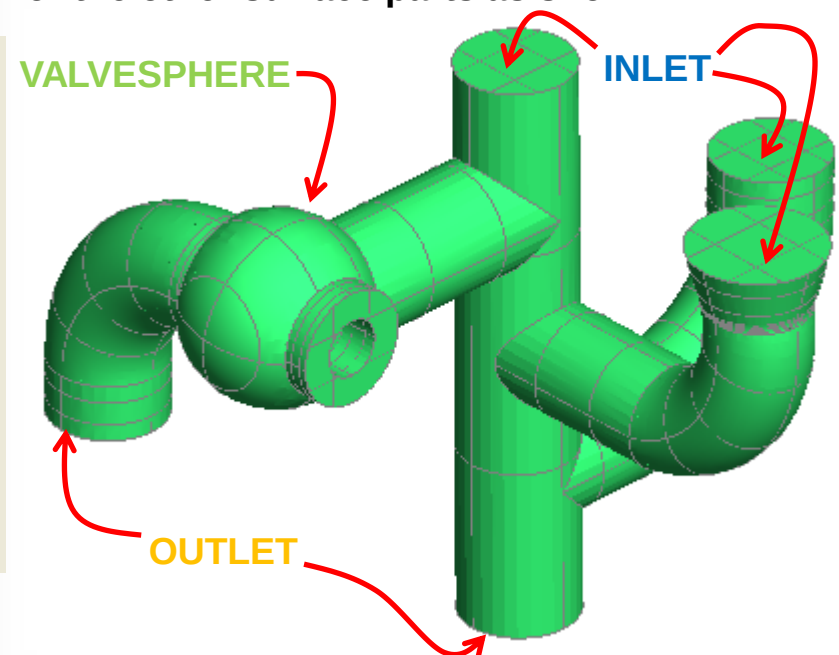
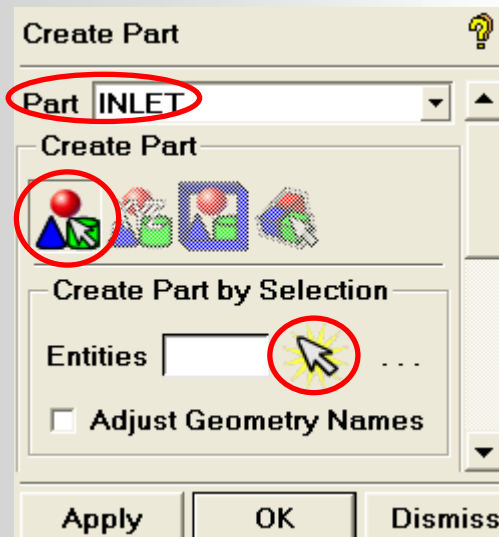
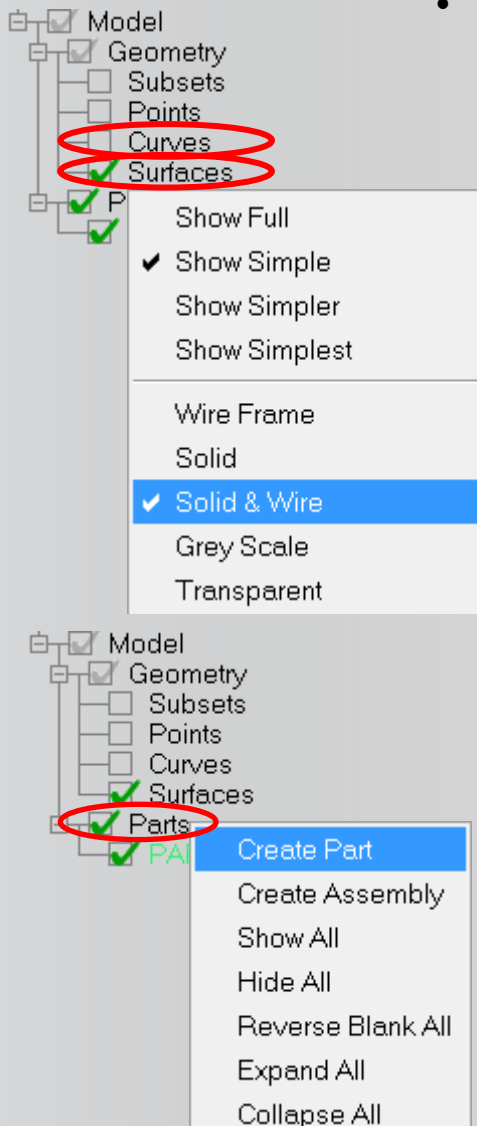
- To Import the Geometry

- **File -> Import Model**
- Set the filter to **Parasolid (*.x_t;*.x_txt;*.xmt_b...)**
- Select: **Valve.x_t** and click **Open**
- Set **Units** to **Millimeter** and click **Apply**



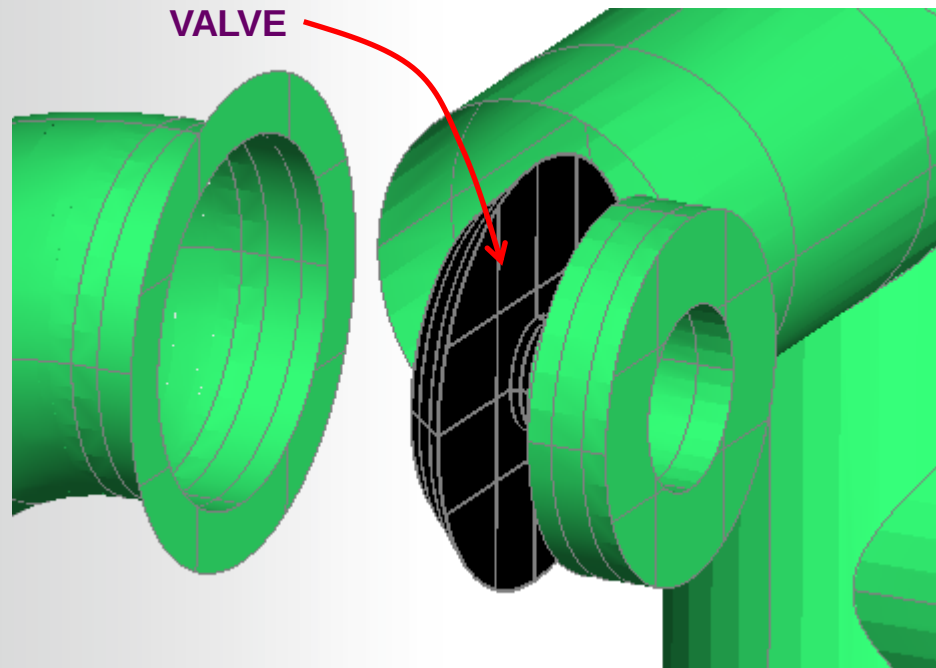
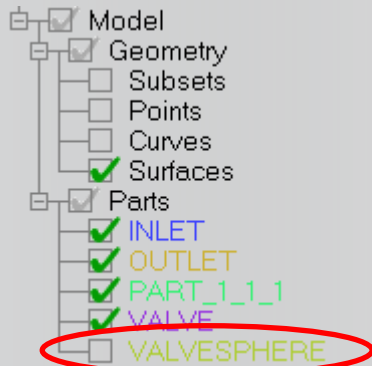
Create Parts

- Separate the Geometry into Parts
 - Activate **Surfaces** by left mouse clicking the check box in the model tree
 - **Right mouse click** in the model tree on **Surfaces > Solid & Wire**
 - Turn off **Curves** visibility to more easily pick surfaces
 - **RMB on Parts** and select **Create Part** from the pull down
 - The **Create Part** panel appears
 - Change the **Part** name to **INLET**
 - Accept the default selection method **Create Part by Selection**
 - **Select** the appropriate 3 inlet surfaces as shown below
 - **Middle mouse click**
 - Repeat the above steps for the other surface parts as shown

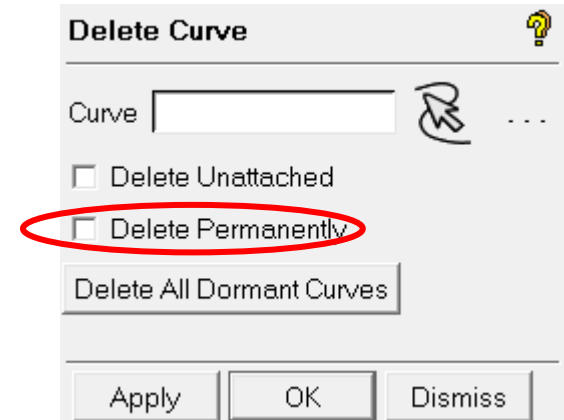
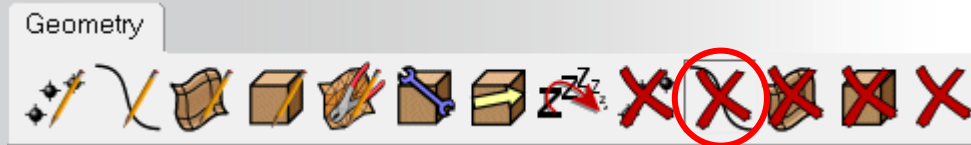


Create Parts – Continued

- Continue dividing the geometry into Parts
 - Turn off the **VALVESPHERE** part in the Model tree. This will expose the Valve Inside
 - Create a new Part, **VALVE**, and select the 5 surfaces of the valve head for this Part
- Separate parts makes it easier to visualize segments of the model by turning parts on and off
- Separate INLET and OUTLET Parts are essential to apply individual boundary conditions. The BC's are applied to the part names.
- Note: the Parts are color matched with their names in the model tree



Delete Curves

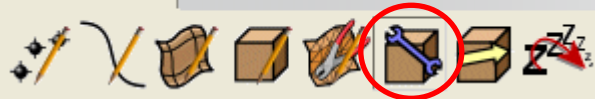


- We want the **Inherit Part** ability within **Build Diagnostic Topology** to assign curves and points to the same parts as the surfaces from which they border
- If the curves are already present, they will remain in the same part, so they must be first deleted
- If curves are deleted non-permanently, they will be made dormant, and if **build topology** determines a curve is needed based on the tolerance and angle filtering, it activates the dormant curve instead of extracting a new curve
- This preserves the more-precise curves from the original CAD geometry instead of extracting new curves
- **Geometry > Delete Curve**
 - Check OFF **Delete Permanently**
 - Turn **curves** ON and **surfaces** OFF
 - Left click, hold, and drag to draw a box over the entire model, then release the left mouse button
 - Middle mouse click



Build Topology

Geometry



Repair Geometry

Part GEOM

☒ Inherit Part

Build Topology

Tolerance 0.1

Filter by angle

Feature angle 30

☐ Filter points☐ Filter curves

Build Topo Parts/Selection

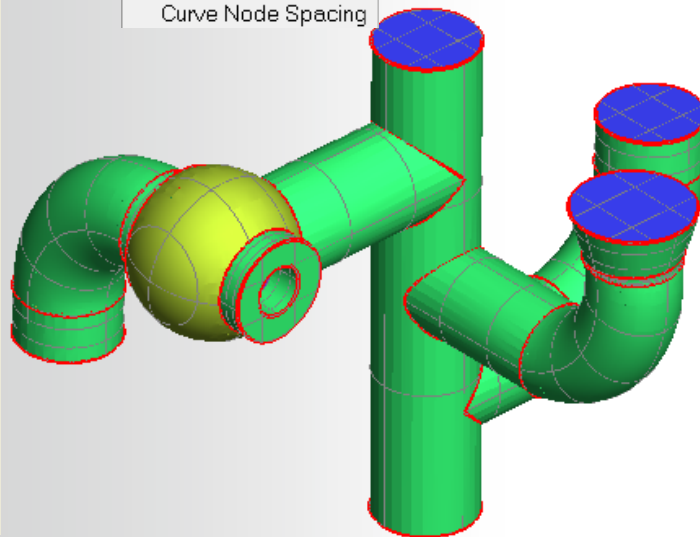
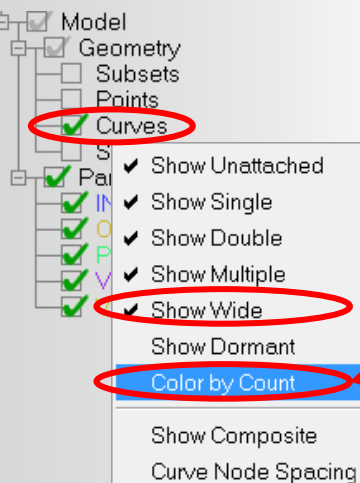
Method All parts

☐ Part by part

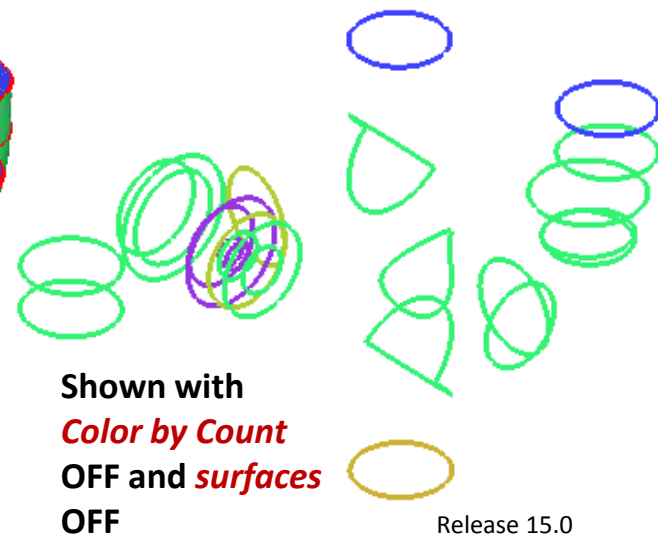
Apply

OK

Dismiss



- **Geometry > Repair Geometry > Build Diagnostic Topology**
 - Accept all defaults
 - Tolerance of 0.1 is computed from diagonal ($1/2500^{\text{th}}$) of bounding box around geometry
 - Ensure that **Inherit Part** is check ON, for the reasons described on the previous page
 - Click **Apply**
 - **Build topology** turns on two curve options: **Show Wide** and **Color by Count**
 - If **Color by Count** is turned off, the curve color will revert to the color of the part

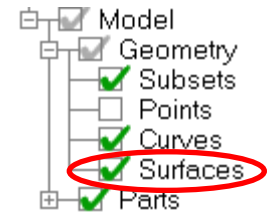
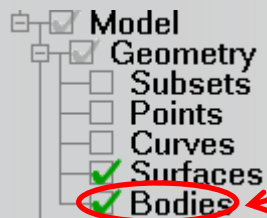
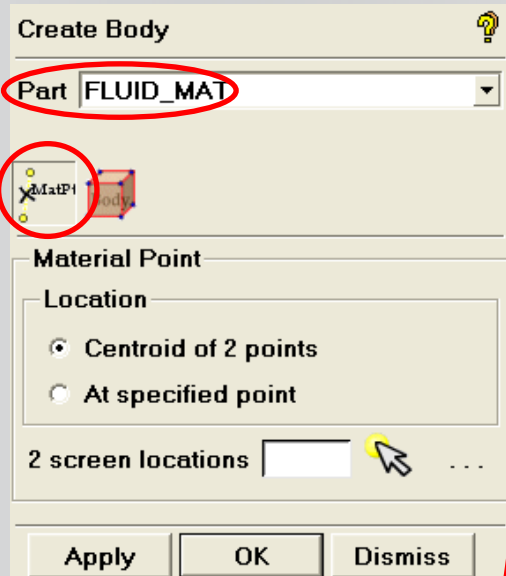


Create Material Point

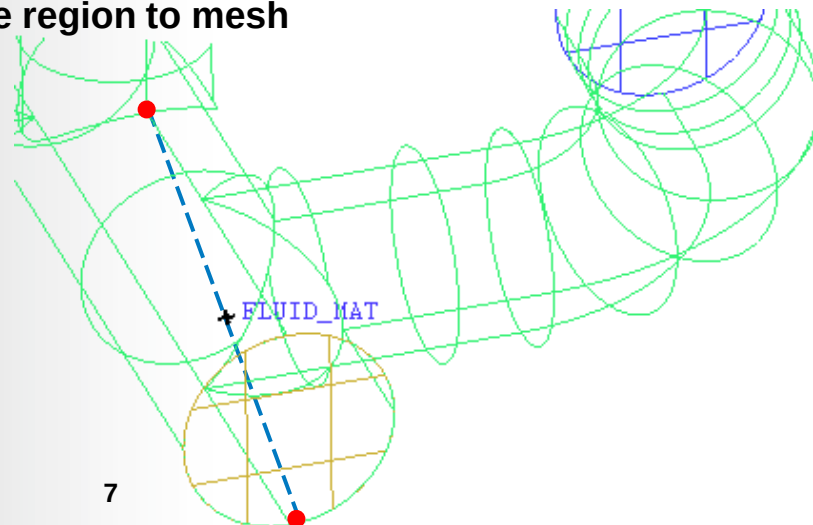


- Tetra Octree will assign the volume elements to the part name of the material point inside the closed collection of surfaces
 - The material point must be inside the volume and not on a surface

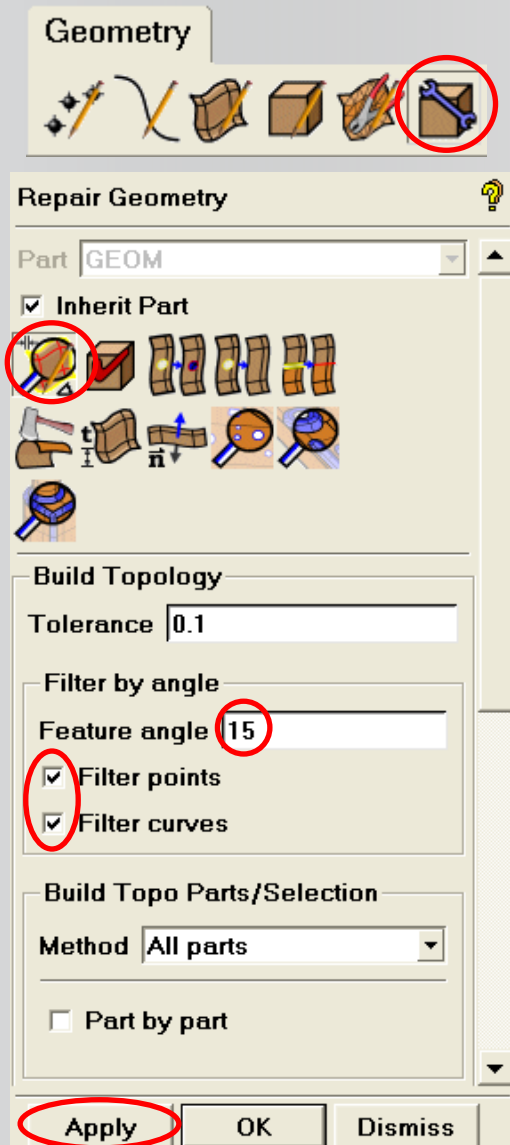
- Select **Geometry > Create Body**



- Turn **surfaces** ON
- Right click on **Surfaces > Wireframe** in model tree
- Enter **FLUID_MAT** as the **Part** name
- Click on **Material Point** button
- Select two screen locations so that the middle of these locations is inside the model
- **Middle click** to accept and then again to exit the function
- Take note that **Bodies** appeared in the model tree
- Rotate the geometry on the screen to ensure the material point is inside the region to mesh

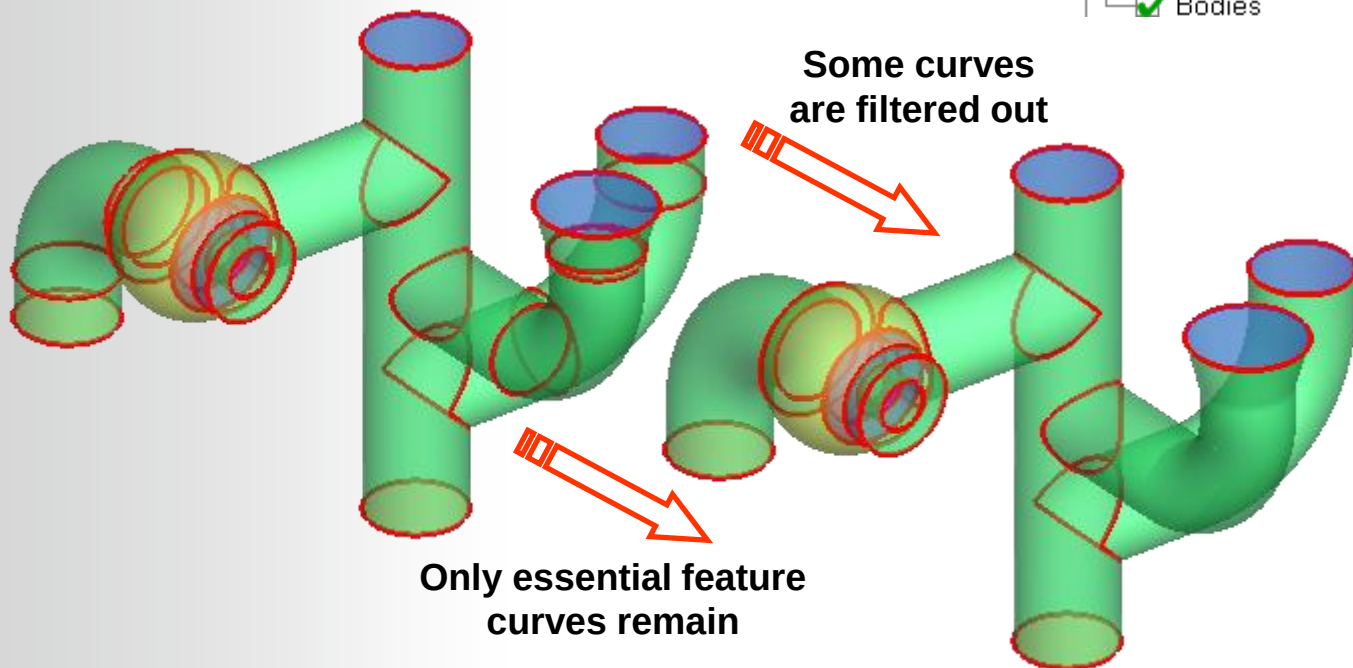
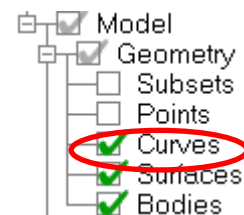


Build Topology



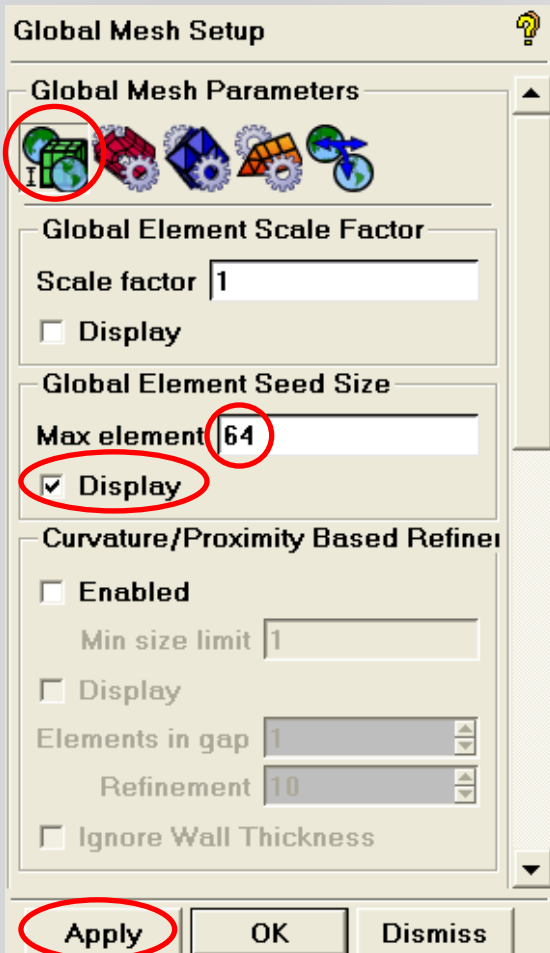
- Remember that curves and points act as constraints to the mesher, so let's remove curves and points which are not at hard angles

- Geometry > Repair Geometry > Build Diagnostic Topology
- Default: **Tolerance** is auto-calculated as 0.1
- Filter by angle: Turn on **Filter points** and **Filter curves**
- Set **Feature angle** to 15
- Default: **Part** name is set to **Inherit Part**
- Turn **Curves** on to see what will change
- Accept other defaults and Click **Apply**

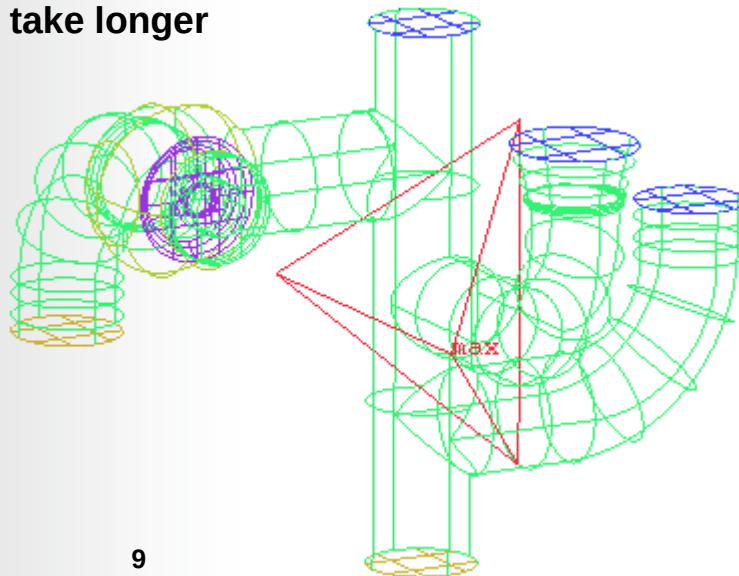


Global Mesh Size

Mesh



- Set the max volume element size in the entire mesh
- **Mesh > Global Mesh Setup > Global Mesh Size**
 - Set **Max element** = 64
 - Click **Apply**
 - Turn on **Display** under **Max element** to see a one-icon display of this size in the center of the model
- This is a very large element size but the mesh will be controlled based on surface sizing performed in the next step
 - The initial surrounding octree mesh is seeded with this size, so too small a max size will make meshing take longer



Surface Mesh Size



Surface Mesh Setup

Surface(s) 

Maximum size

Height

Height ratio

Num. of layers

Tetra width

Tetra size ratio

Min size limit

Max deviation

Mesh type

Mesh method

☐ Remesh selected surfaces

Blank surfaces with params

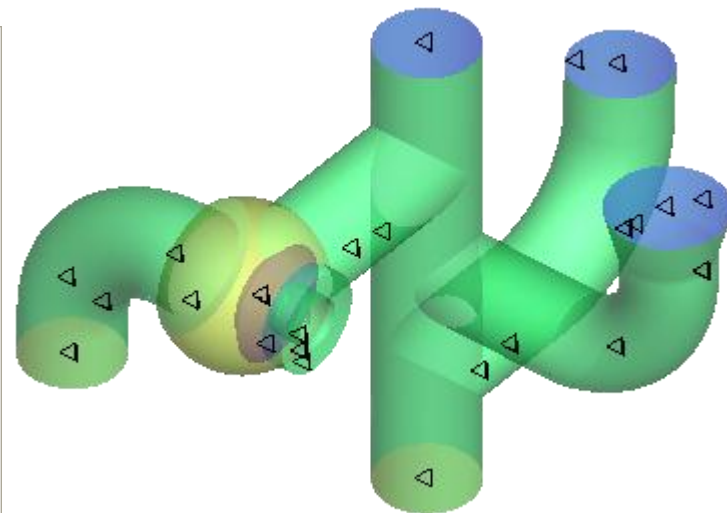
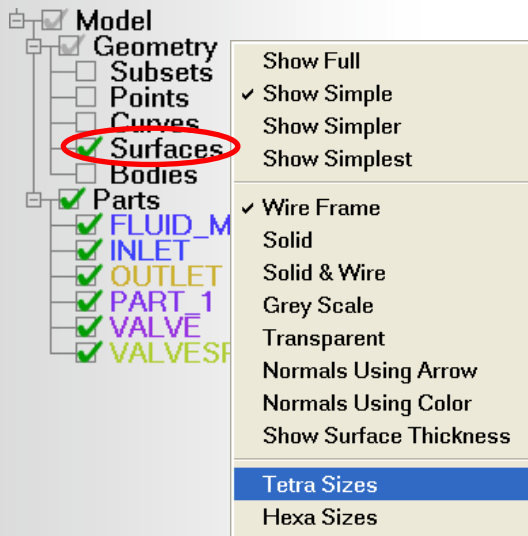
Apply OK Dismiss

- Set Surface Mesh Size

- **Mesh -> Surface Mesh Setup**
- Click the **Select surface(s)** picker 
- The **Select geometry** popup tool bar appears, use the **Select All** icon to select all the surfaces

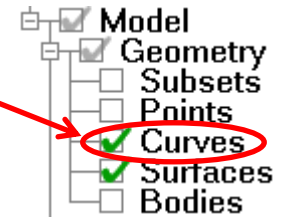


- Set the **Maximum size** to 4
- Click **Apply**
- **Right mouse click** on **Surfaces > Tetra Sizes** in Model tree to see a visual display of the sizes

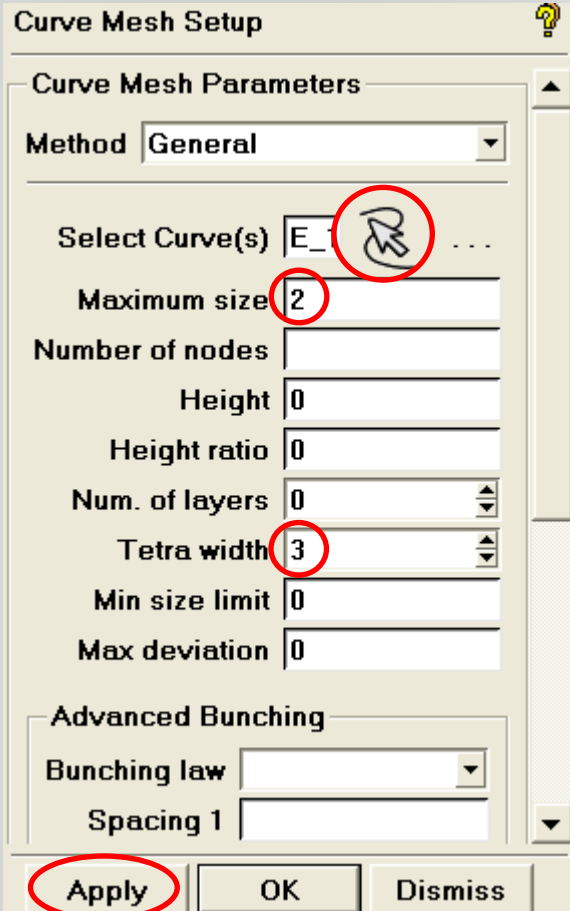


Curve Mesh Size

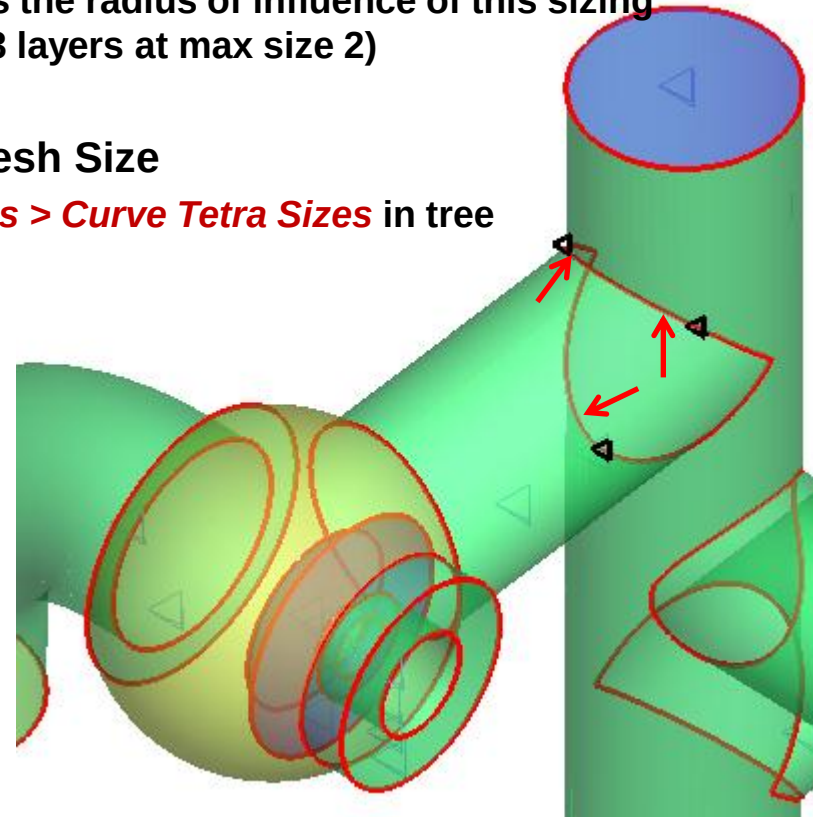
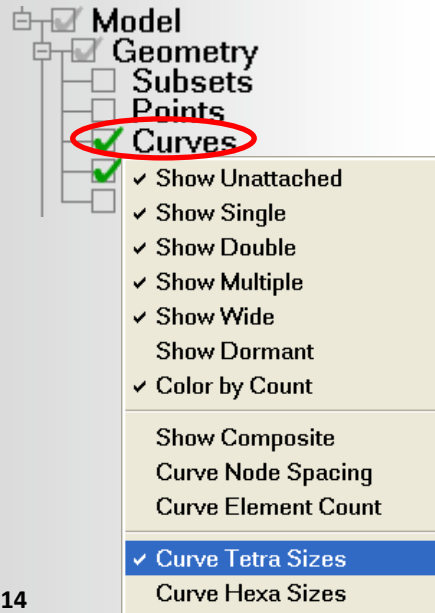
First turn on **Curves** in the model tree



Mesh



- Go to **Mesh -> Curve Mesh Set up**
 - Method -> General**
 - Press the **Select Curve(s)** button  and select the indicated 3 curves, then **middle mouse**
 - Set **Maximum Size** to **2**
 - Set **Tetra Width** to **3**
 - Increases the radius of influence of this sizing control (3 layers at max size 2)
 - Click **Apply**
- Display Curve Mesh Size
 - RMB on Curves > Curve Tetra Sizes** in tree



Create Density Region



Create Density

Name: density.1

Size: **2**

Ratio: 0

Width: 0

Density Location

From:

☐ Points ☒ **Entity bounds**

Entities:

Apply OK Dismiss

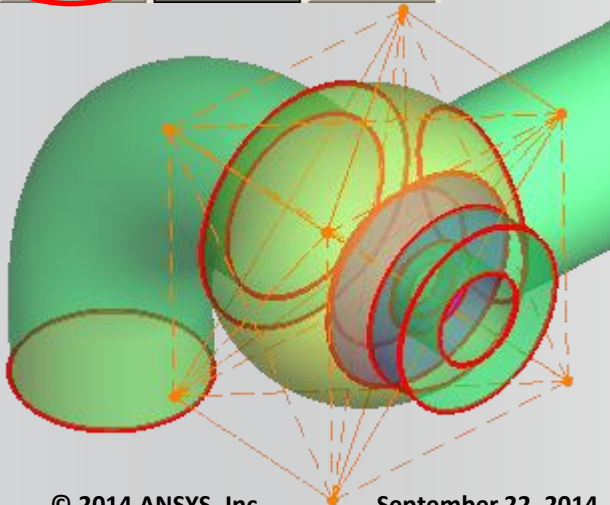
- Create Density Box to locally control volume mesh size
 - Surface and curve sizes only control mesh sizes on those entities. Densities control mesh sizes within a volume.
 - **Mesh -> Create Mesh Density**
 - Set the **Size** to **2**
 - Set the **Type** to **Entity bounds**
 - Use the **Select geometry** picker
 - The **Select geometry** popup tool bar appears, use the **Select Items in a Part** icon to select the **ValveSphere** Part
 - **Accept**, then **Apply**



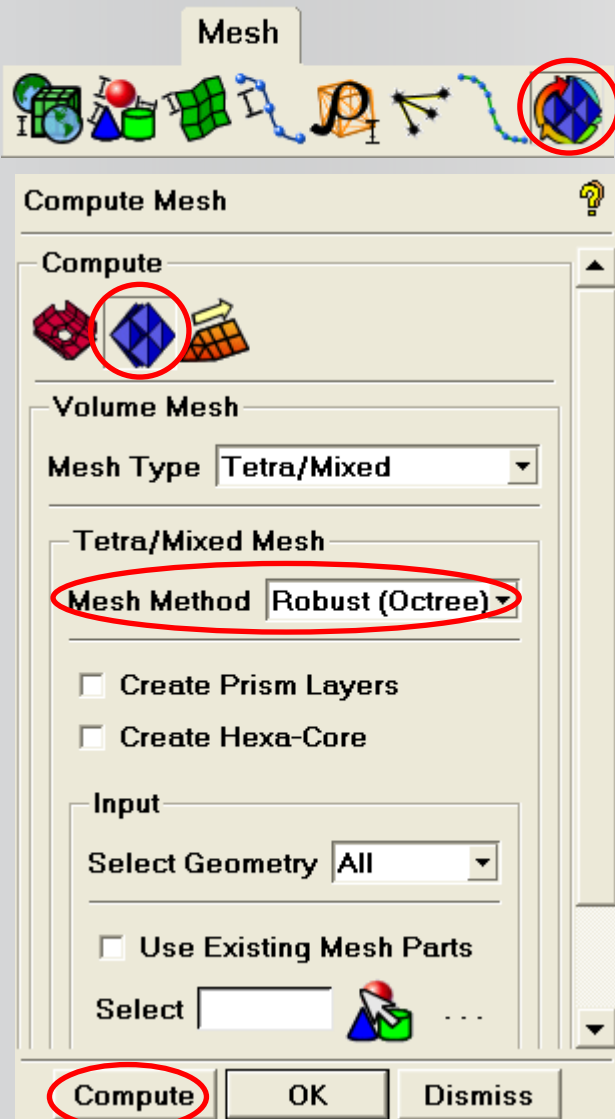
Select part

☐ FLUID_MAT	☐ OUTLET	☐ VALVE
☐ INLET	☐ PART_1_1_1	☒ VALVESPHERE

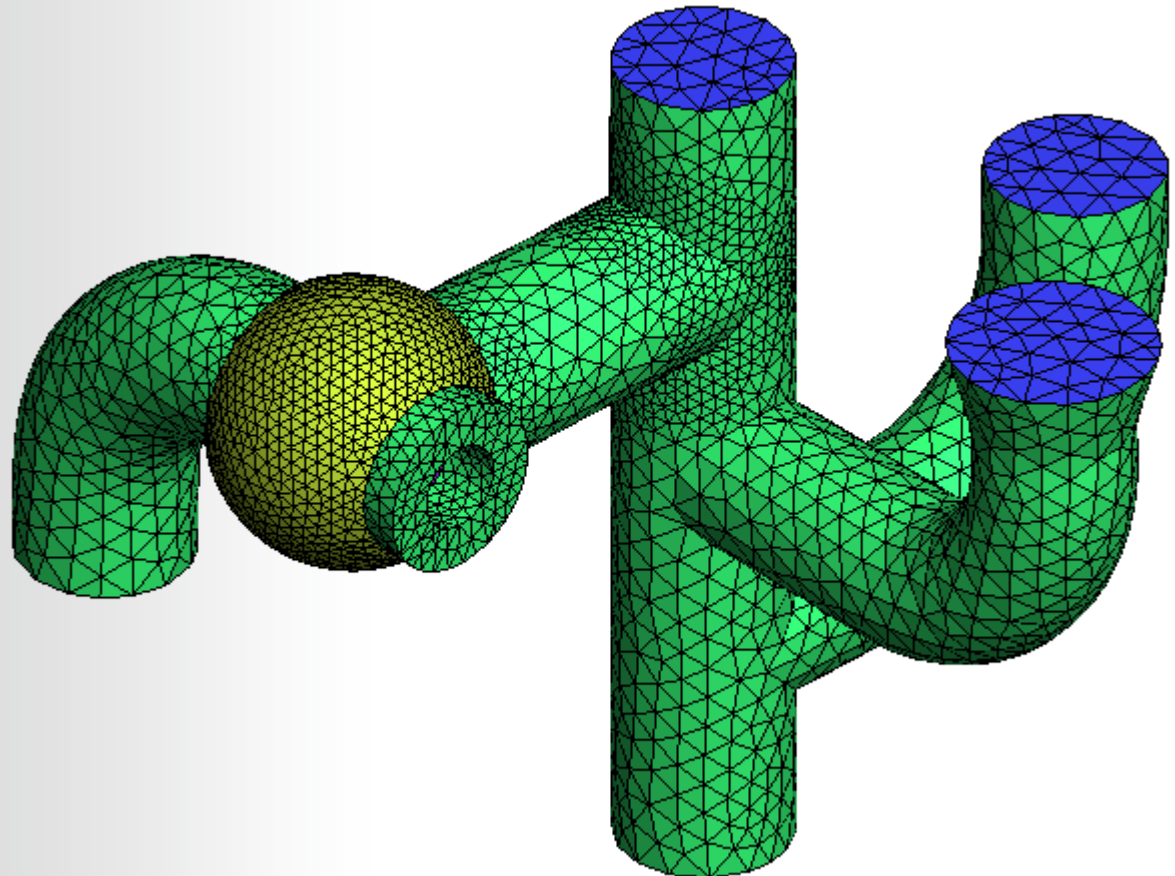
Accept All None Screen Cancel



Compute Mesh



- Select **Mesh > Compute Mesh > Volume Mesh**
 - Set **Mesh Method > Robust (Octree)** (default)
 - Click **Compute**



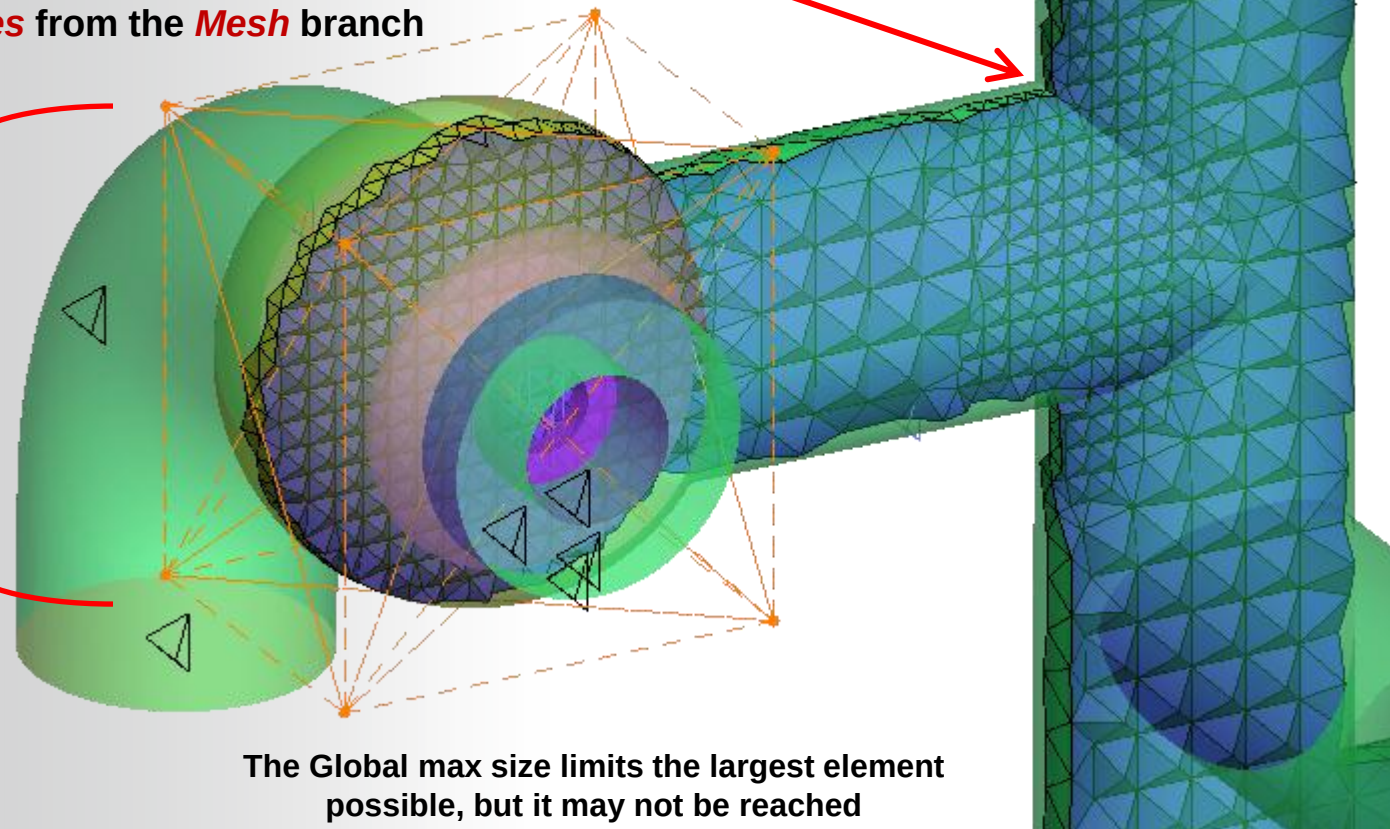
Cutplane View

- Cutplane View
 - **RMB on Mesh > Cut Plane... > Manage Cut Plane** in the model tree
 - Change the Cut Plane **Method** to **Middle X Plane**
 - Adjust the **Fraction Value** to display this image (about 0.43)
 - Activate **Volumes** from the **Mesh** branch

The Surface Tetra Sizes affect the mesh near the surface

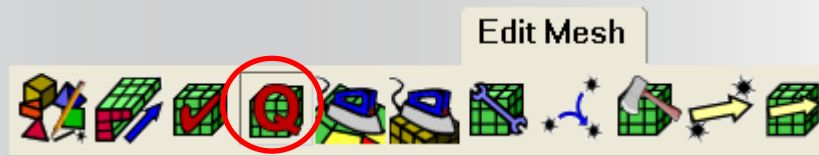
The Curve Size (2) and width set to 3 affect the mesh size within at least three elements of the curves

The Density region controls the max element size within the valve sphere



The Global max size limits the largest element possible, but it may not be reached

Check Mesh Quality



Quality Metrics

Mesh types to check

	Yes	No
LINE_2	☐	☒
TETRA_4	☒	☐
TRI_3	☒	☐

Elements to check

☒ All ☐ Active parts
☐ Visible subsets ☐ Visible subsets and active parts

Refresh Histogram

Quality type

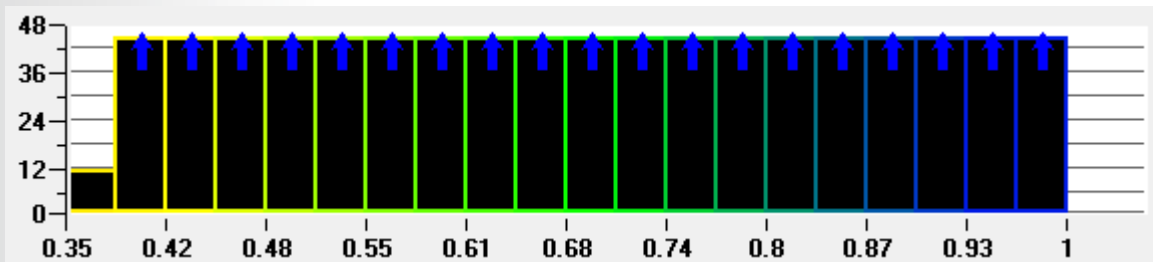
Criterion Quality

Apply OK Dismiss

- Check mesh quality
 - Select **Edit Mesh > Display Mesh Quality**
 - Set the **Criterion** to **Quality** (default)
 - Click **Apply**
- Check the quality histogram. Smooth mesh if needed (everything above 0.2)

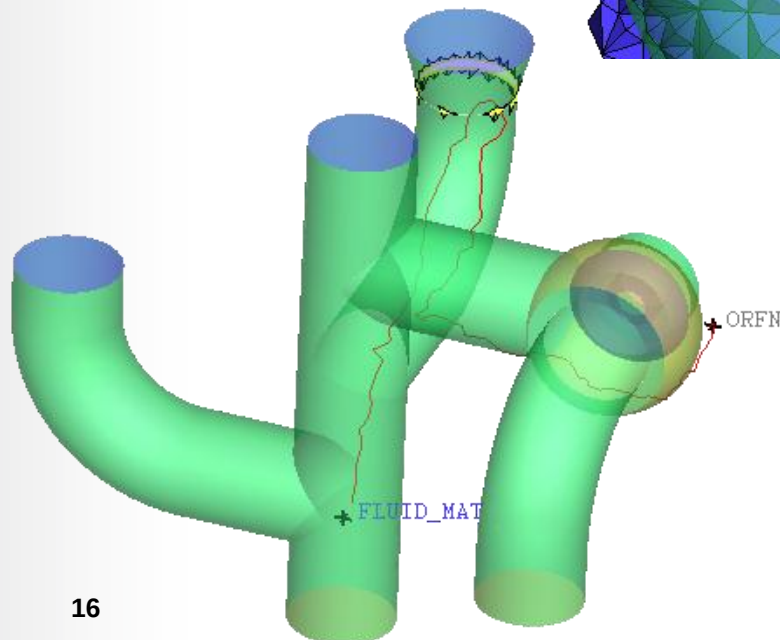
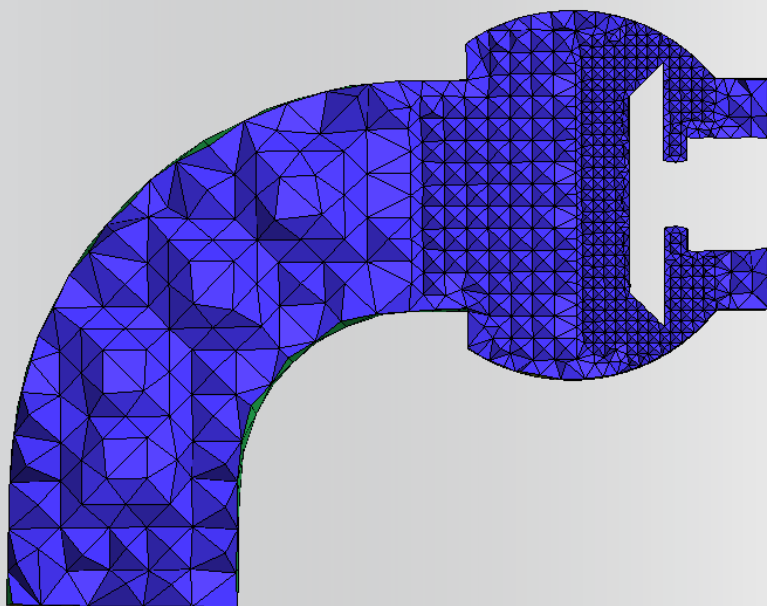
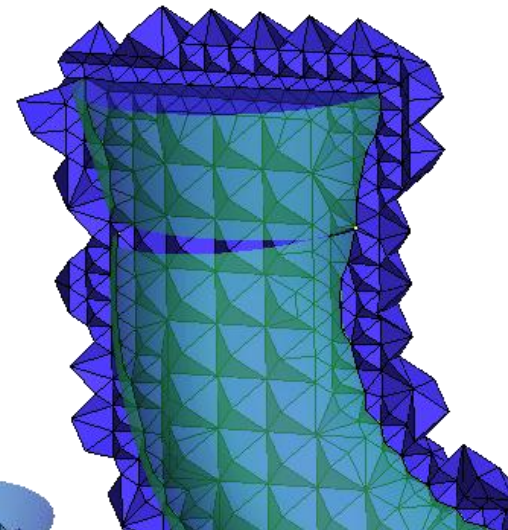
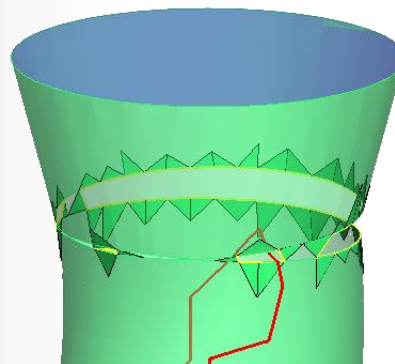
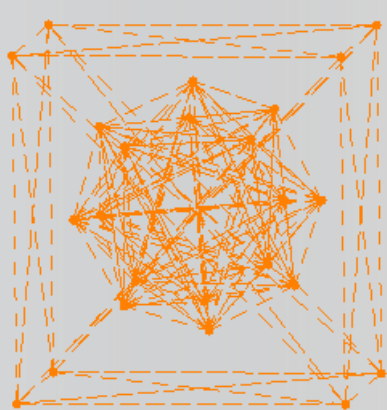


- Save Project when satisfied with quality
 - **File > Save Project as...**
 - Enter **Valve.prj**



Further Investigation

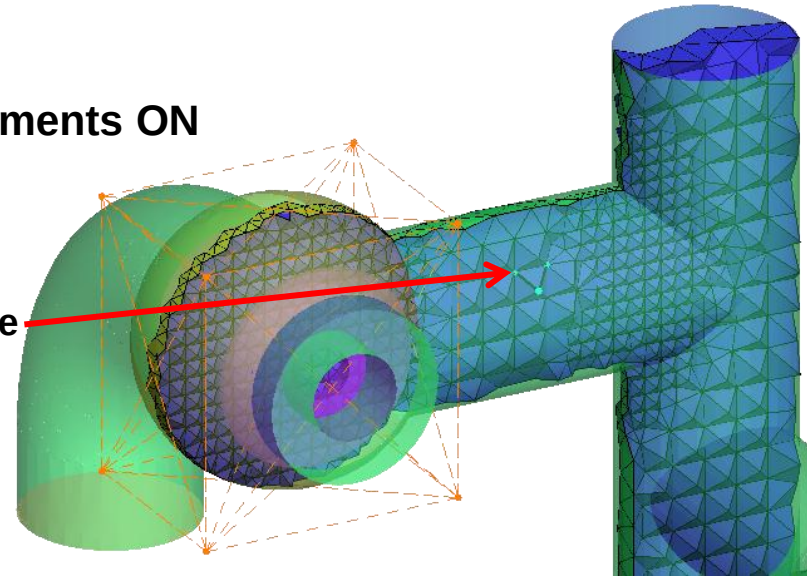
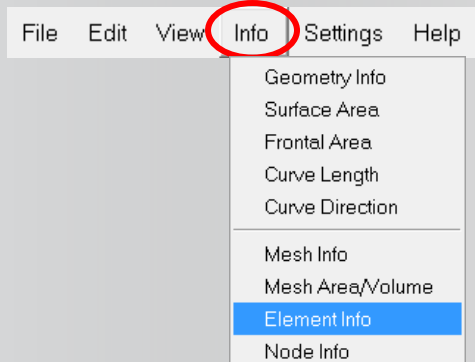
- For those that have the time, the following pages introduce additional concepts



- First, go into cutplane mode with volume elements ON

- **Info > Element Info**

- Use to get information on individual elements
- Select one volume element inside the cutplane
- Middle mouse click

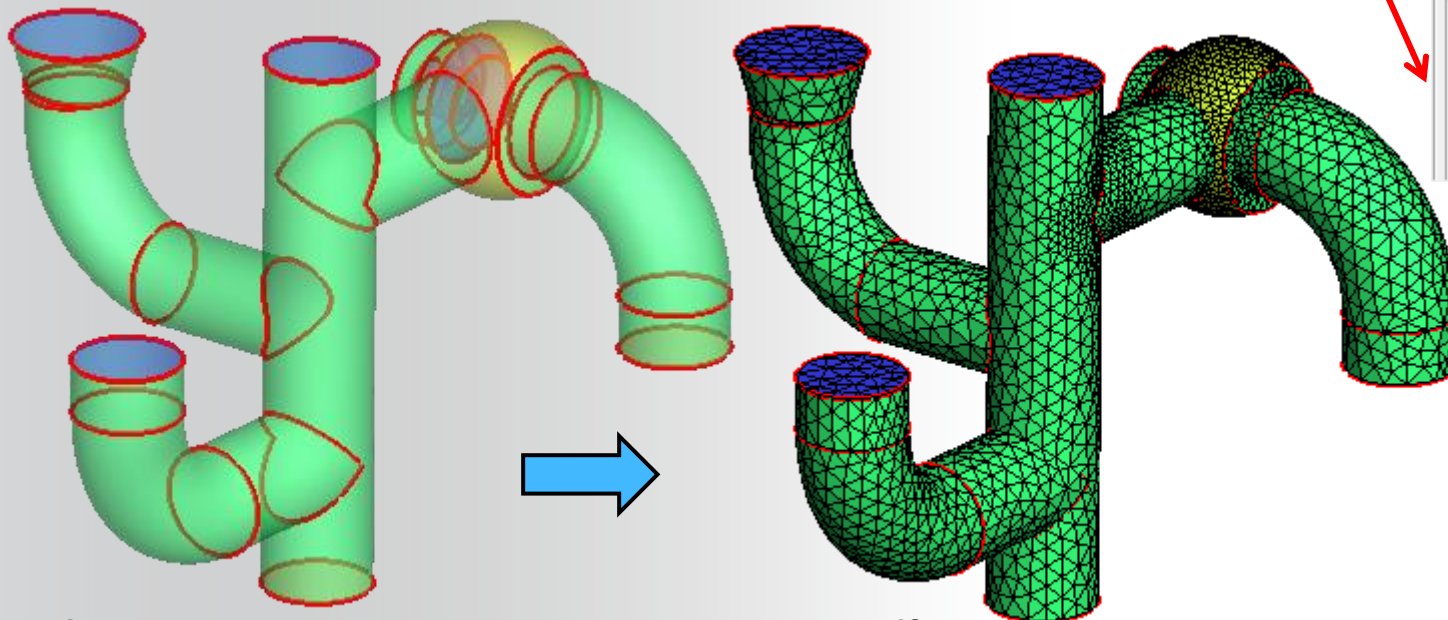
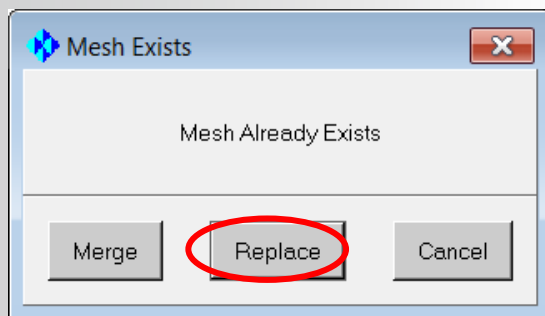


TETRA_4 15999 (internal 8232) **part FLUID_MAT** nodes (internal 4185 4189 12037 4022)
 Mesh Quality
 Min = 0.929516, max = 0.929516, mean = 0.929516
 1 elements with the "Quality" diagnostic

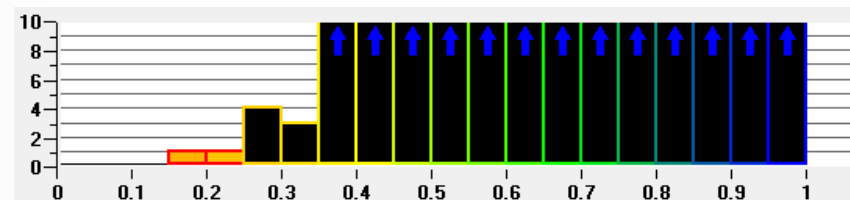
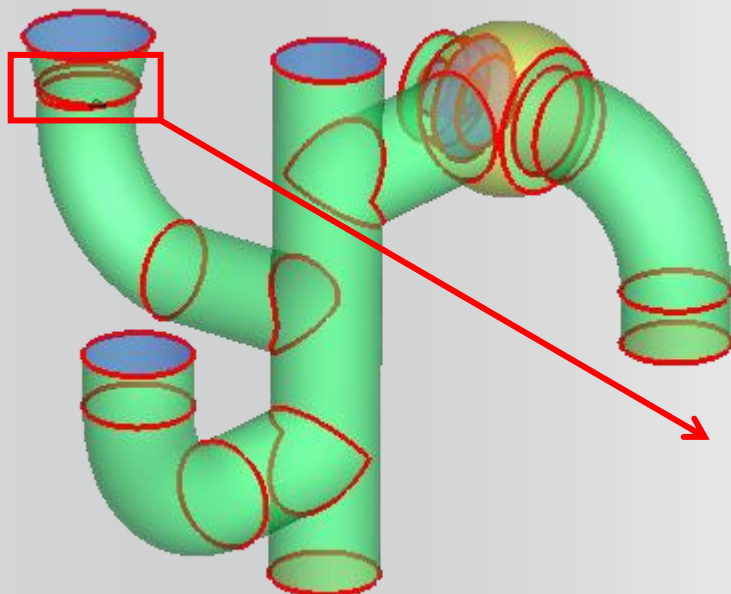
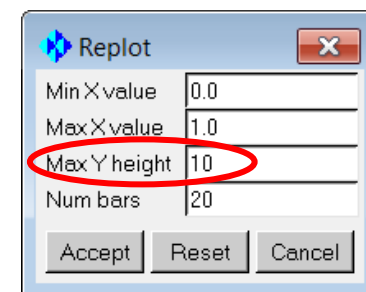
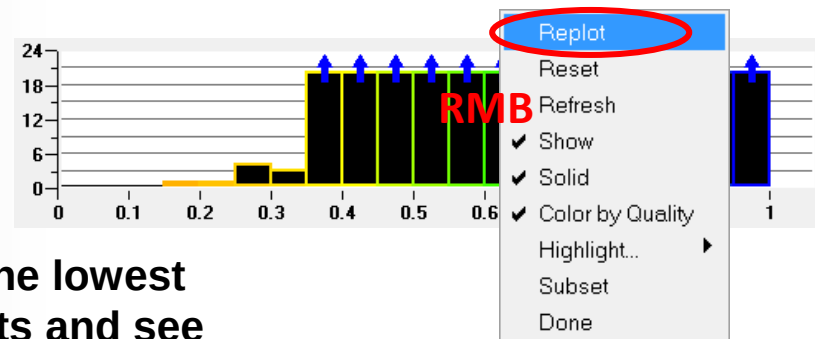
- Most often, it is the part grouping that is the desired information
 - The volume element is in the part of the material point, FLUID_MAT
 - If no material point was created, it would be in CREATED_MATERIAL_<number>
 - This grouping name gets translated to the solver as the tag which solid or fluid properties are set
 - Fluent = “zone”, CFX = “3D principal region”, Ansys = “nodal/element component”

Octree Mesh with More Constraints

- **Build Diagnostic Topology** again with filtering off
- Run **Tetra Octree** on this modified geometry
 - Choose to **Replace** mesh if you have not previously closed the current mesh



- Smooth the mesh the best you can
- You would like to be able to select on the lowest histogram bars to highlight the elements and see where the problems are at, but it is hard to see the smallest bars or select them
 - Right mouse click on the histogram and choose **Replot**
 - Specify a small **Max Y height**, such as 10
 - You can now see and select the lowest histogram bars

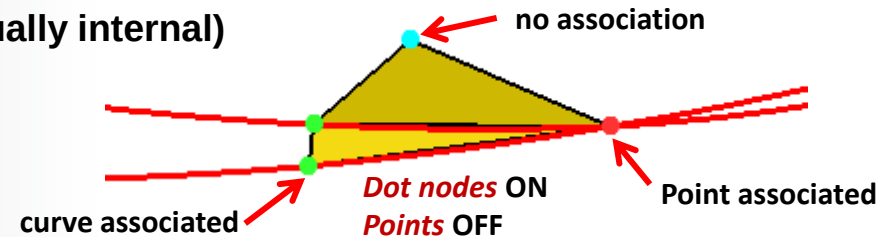
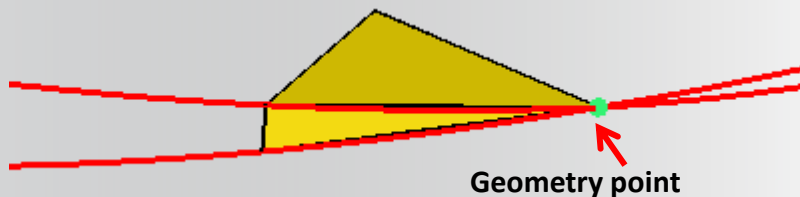
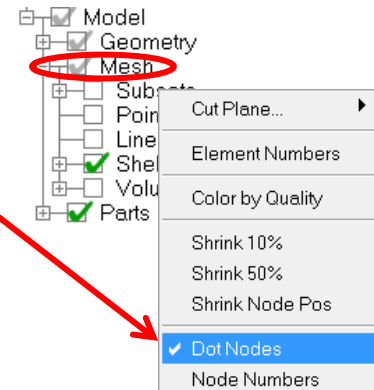


Mesh Constraints

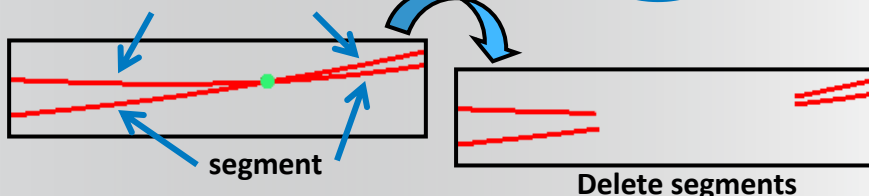
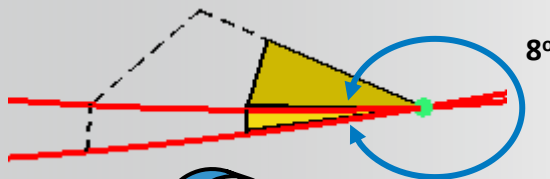
- This element won't smooth any better because of the additional curve constraints

- You can turn on **Dot Nodes** in the model tree to see a color coding for node constraints

- Green node = curve constrained (nearest curve)
- Red node = point constrained
- Black or white node = surface constrained (nearest surface)
- Blue node = no constraints (usually internal)



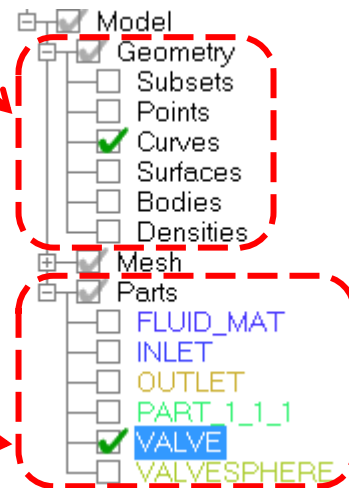
- For most problems, setting the local element size smaller (on surfaces, curves, or global min size limit) will allow extra geometry constraints to be modeled with decent quality
- But in this case, the low angle stays about the same even with a smaller mesh size



- This is why it is necessary to remove the low-angle curve constraints
- This can be done by running **build topology** with **"filtering"** because the surfaces are tangent, or by segmenting curves and deleting the small angle
- It can be fixed at the mesh level by merging nodes, either interactively, or possibly all within a tolerance

Mesh Density – Polygon Shaped

- First turn OFF all geometry items except **Curves**
- Turn OFF all parts except **VALVE**
 - You'll be left with just the curves bordering the valve surfaces (because of *build topology* with “*inherit part*”)
- Create a **Mesh Density** using the **Points** method
 - Set a size of 1, which is half the surrounding density size



Create Density

Name: density.1

Size: 1

Ratio: 0

Width: 0

Density Location

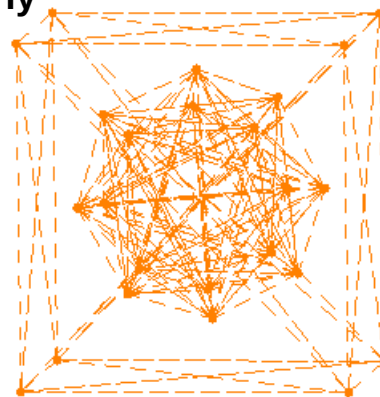
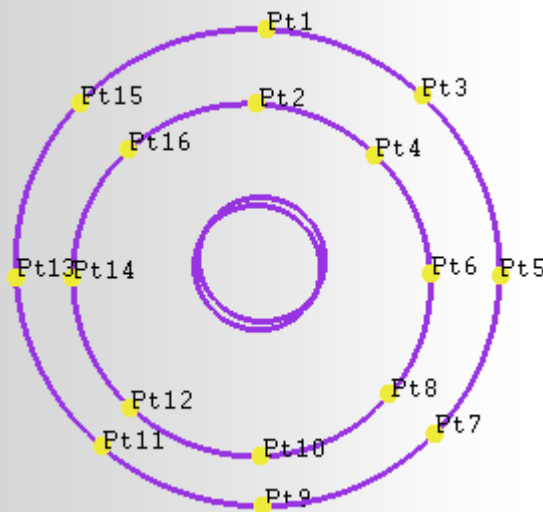
From:

☒ Points ☐ Entity bounds

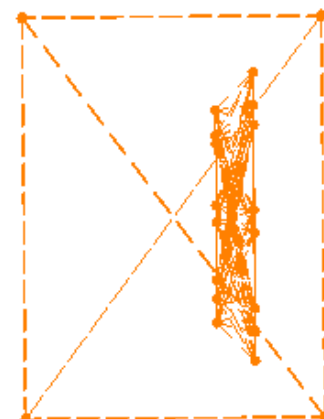
Points: {5.9999991}

Apply OK Dismiss

- Select on many locations on the outer two curves of the valve as shown (order doesn't matter)
- Turn ON **Densities** and all parts after creating the density and turn OFF **Curves** to see it more clearly



Density shown inside the previous density



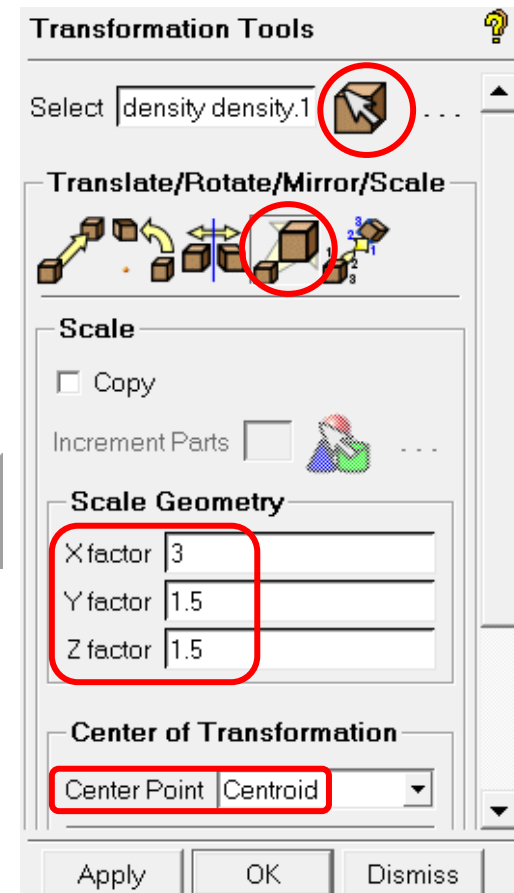
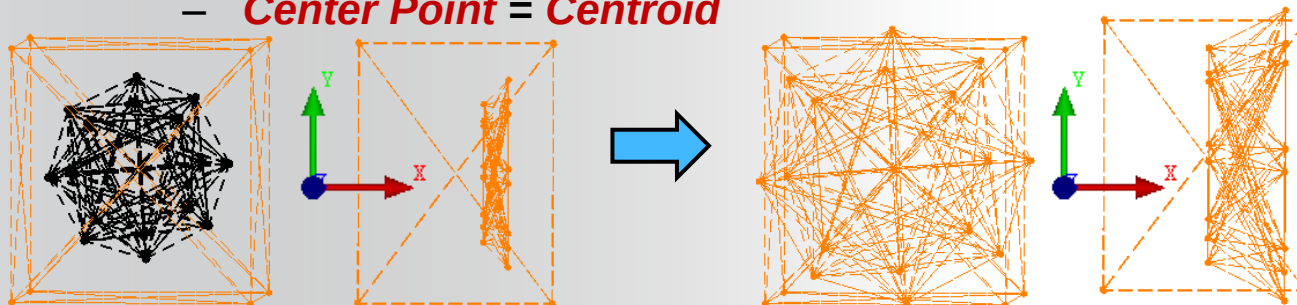
View from side

Scale Mesh Density

- Scale the density larger
 - Geometry > Transform Geometry > Scale Geometry**
- Often the most bothersome task of creating a density is establishing the locations for the density corners since they are typically not on geometry entities
- So it may be easier to create the density directly on geometry and then scale it larger or move it
 - Press the **Select geometry** button  and select the inner density
 - Turn off point, curve, and surface selection if you have trouble selecting the density



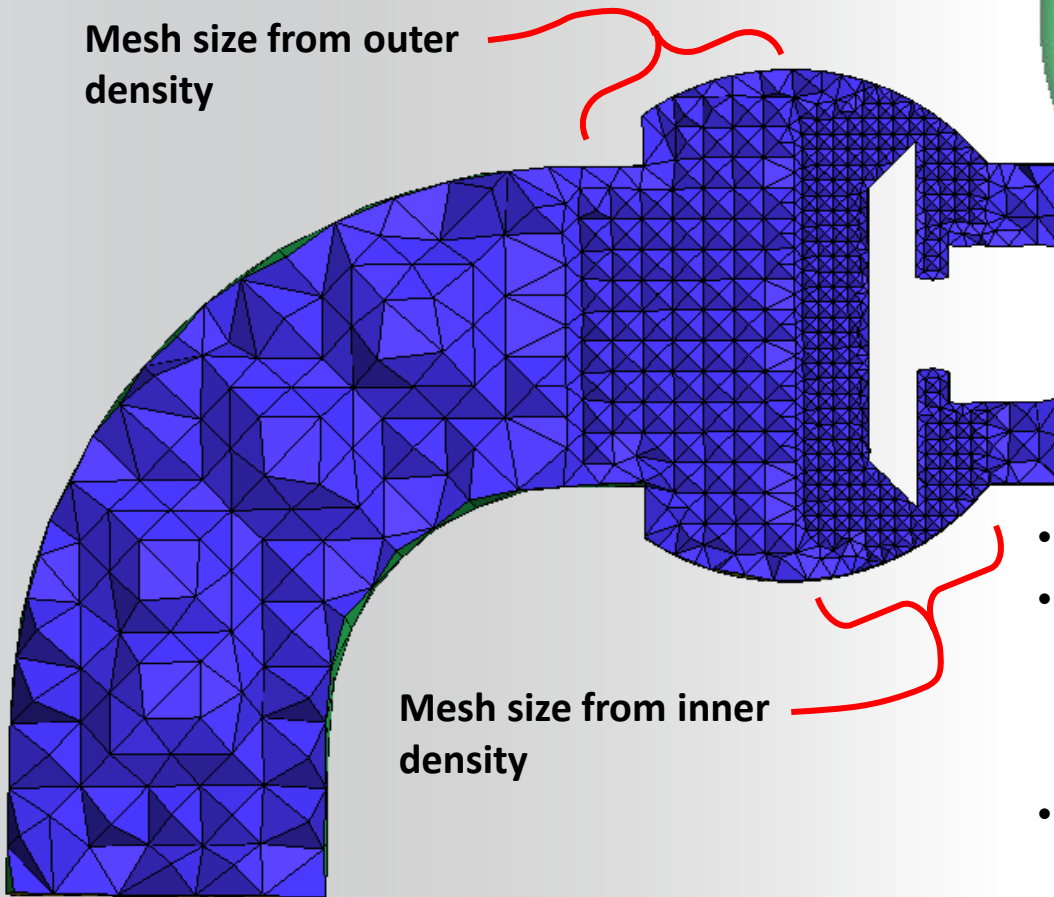
- X factor = 3, Y factor = 1.5, Z factor = 1.5**
- Center Point = Centroid**



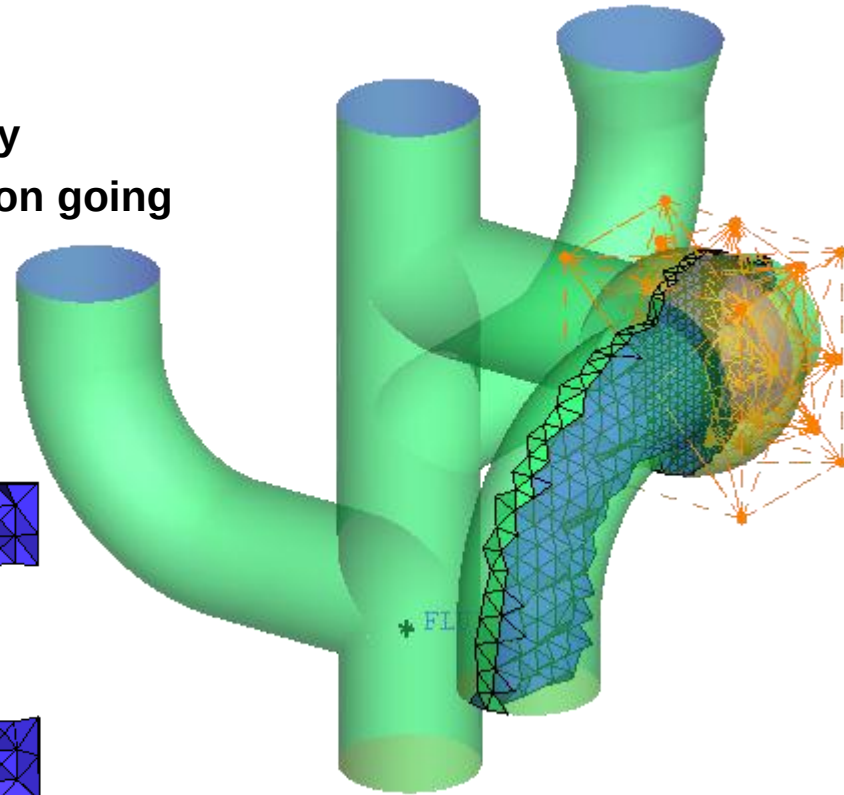
Inspect Mesh

- Generate an Octree mesh on the geometry
- Inspect a cutplane normal to the Z direction going through the VALVESPHERE

Mesh size from outer density



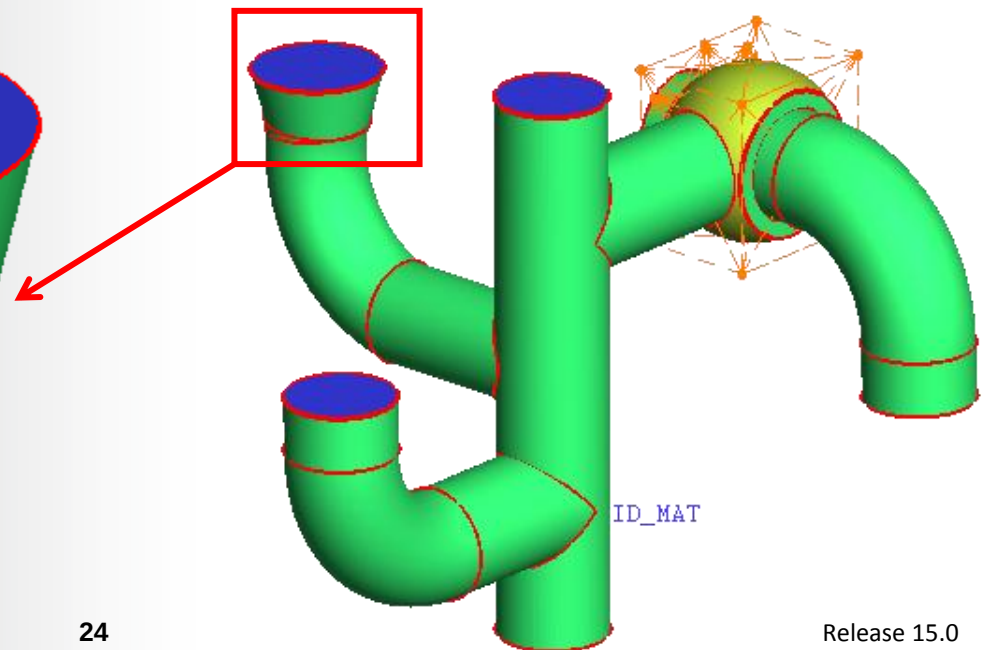
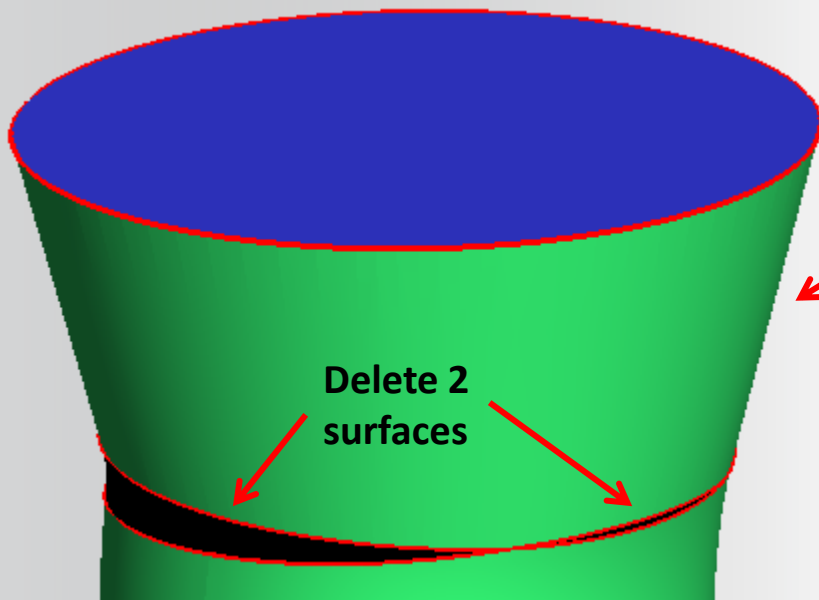
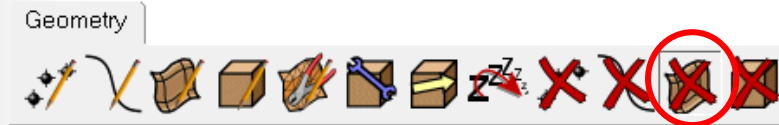
Mesh size from inner density



- This illustrates an important concept
- Multiple mesh sizes can be prescribed in any given location due to global setup, surface/curve sizes, part mesh sizes, densities, etc...
- The smallest size is always favored in any given location

Leakage – Delete Surfaces

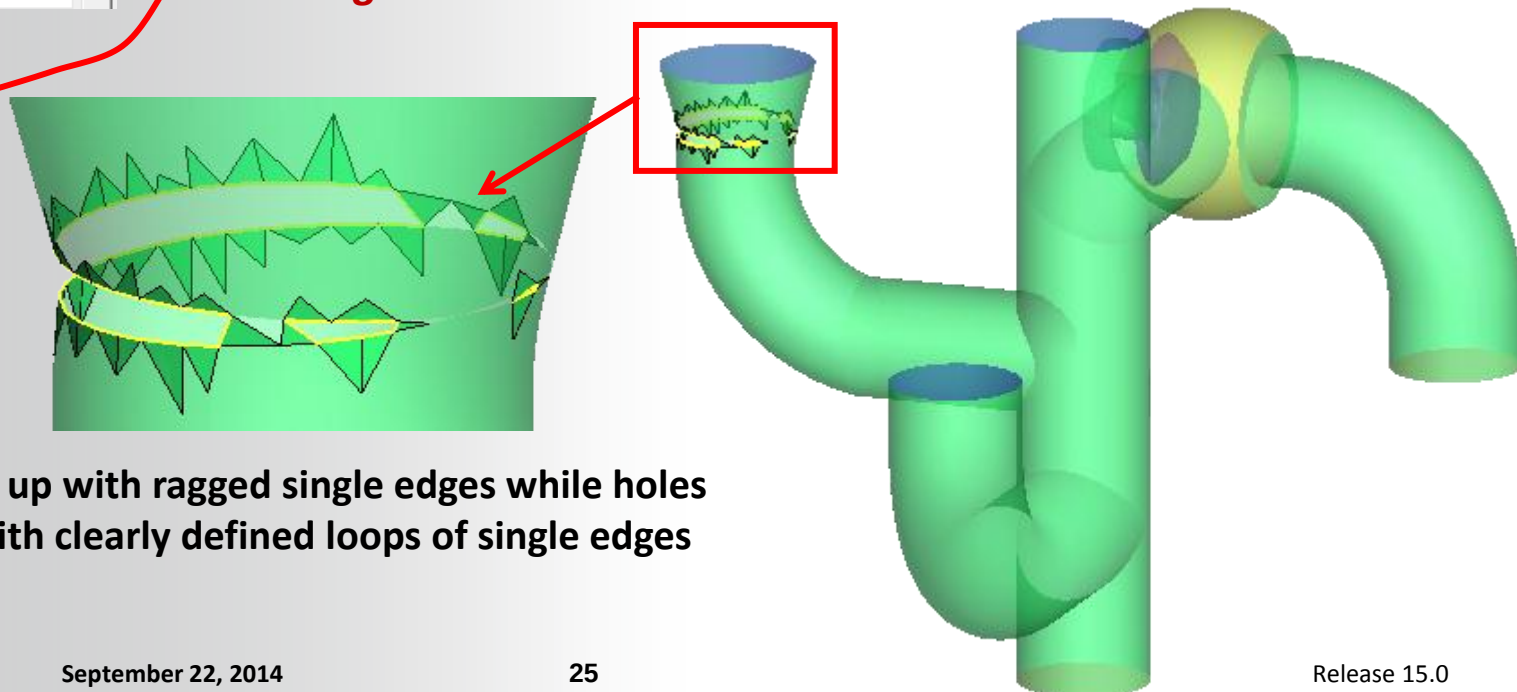
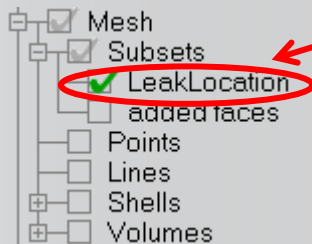
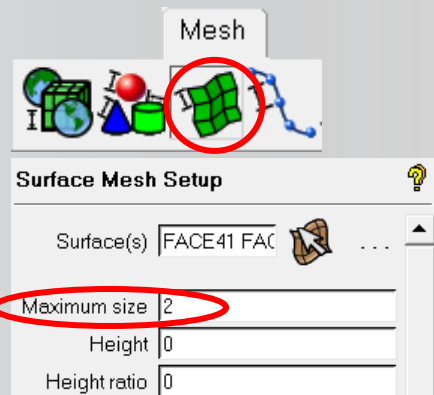
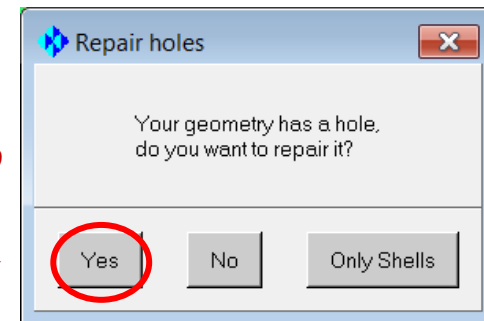
- “Leakage” can occur with tetra octree during the last third of the meshing process when it attempts to mark tetra elements into the parts of the material points (“*running cutter*”)
- If it cannot find enclosed surface regions around material points, it cannot figure out which elements to mark into certain parts and which elements to delete
- In order to demonstrate this, you must first create some gaps or holes
- **Geometry > Delete Surface**
 - Select the 2 surfaces shown



- Before Octree meshing, set the max size on all surfaces to half (2) of the current size (4)

- Generate the Octree mesh

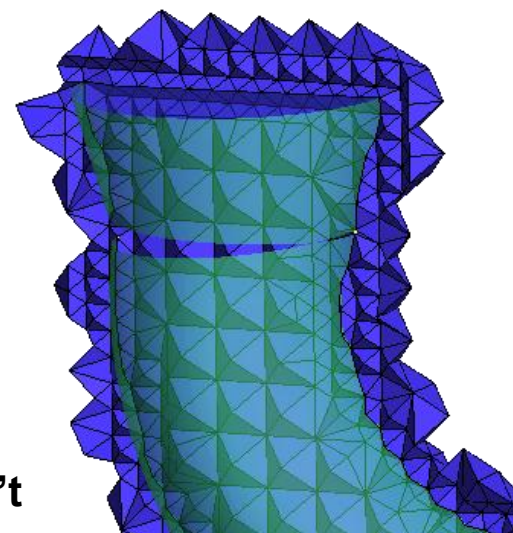
- A popup message will read, “*Your geometry has a hole, do you want to repair it?*” Press **Yes**
- This will create a subset called **Leak Location** which displays the *single edges*



Gaps show up with ragged single edges while holes show up with clearly defined loops of single edges

Create ORFN Material

- A red jagged line may be present which shows the leakage path between material points when multiple material points are present
- Create an **ORFN** material point in the tetras to be cut away
 - **ORFN** is a reserved name for the region to delete
 - A cut plane shows tetras outside the geometry, but if you create the **ORFN** point outside of any tetras, it won't work, so look in the messages for the location where octree creates it and create one at the same place



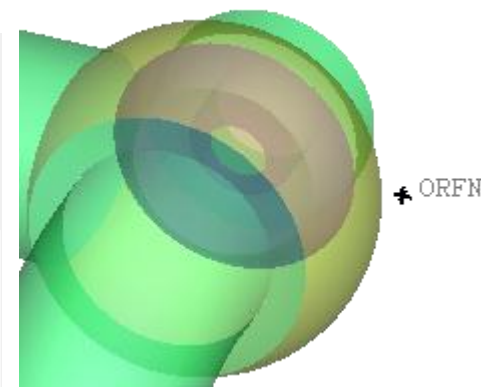
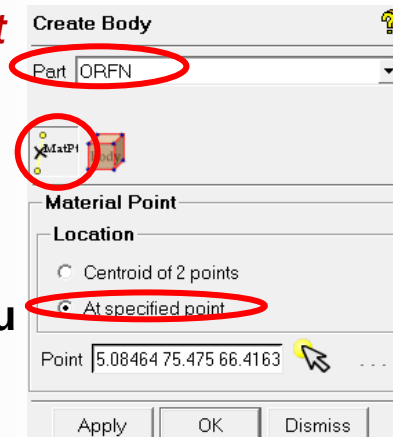
Cutplane

running cutter
computing opaque faces
adding missing faces
0 faces added

Adding new material point ORFN/-1 outside object at (5.08464 75.475 66.4163)
determining material point containment
Performing flood fill

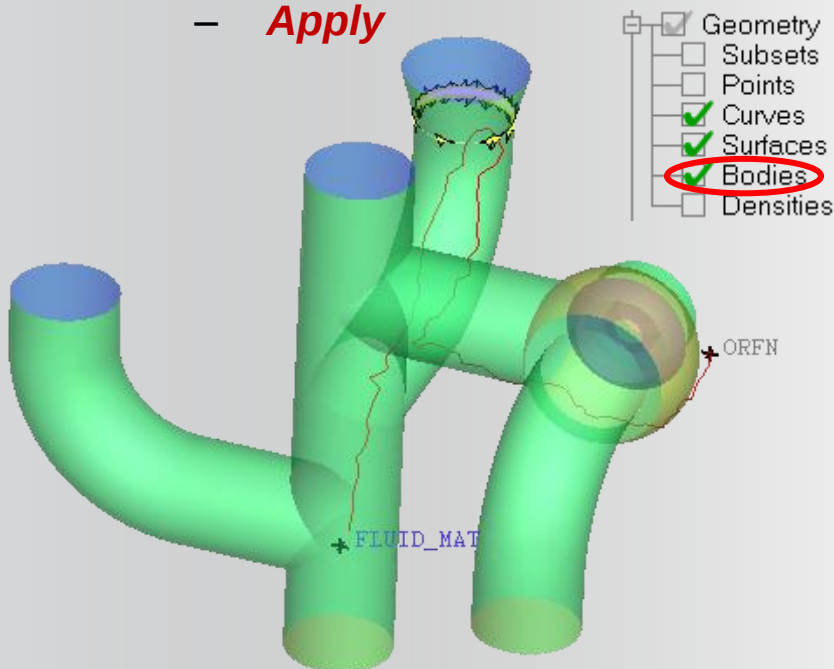


- **Geometry > Create Body > Material Point**
 - Part = **ORFN**
 - Location = **At specified point**
 - Highlight the coordinates in the message window and use **Ctrl-C** to copy and **Ctrl-V** to paste in the menu
 - **Apply**

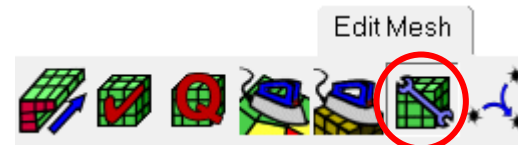
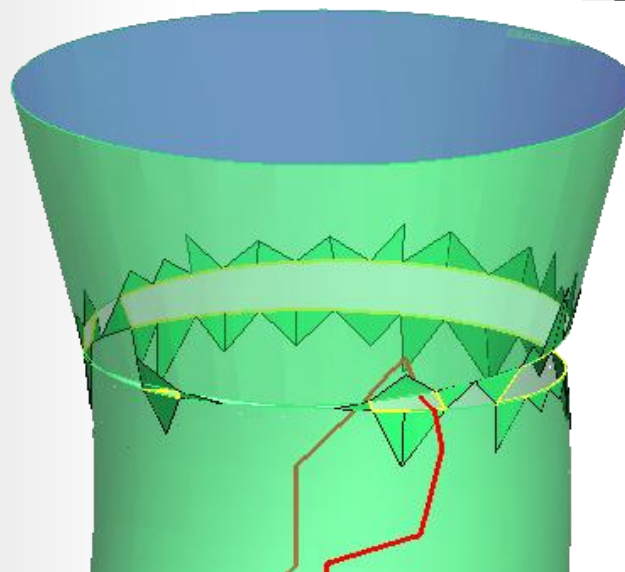


Red Jagged Line

- If you were to remesh with 2 material points present, you would see a red jagged line connecting the material points if there is leakage between the regions
- But you don't have to perform the entire meshing operation over again. You can just perform the last step called "**Flood Fill**," which determines the containment
- **Edit Mesh > Repair Mesh > Flood Fill/Make Consistent**
 - Turn on only **Flood Fill**
 - **Apply**



- Turn on **bodies**



Follow the red jagged line from **ORFN** until it looks like it is passing into the model. This is the location of the leakage. Turn on surfaces to inspect.

Methods to Fix

- There are two ways to fix leakage
 1. Repair the geometry and remesh
 - Usually just creating fill surfaces, but may require more complicated repairs
 2. Repair the mesh
 - Surface mesh the hole(s)
 - **Make consistent** will adjust the tetra mesh to match the repaired surface mesh
 - Octree meshing will have the leakage again
- **Edit Mesh > Repair Mesh > Mesh from Edges**
 - Box or polygon select each loop of single edges one-at-a-time and middle click or **Apply**
- If **Keep volume consistent** is ON, the tetra mesh will be adjusted after each loop is meshed
- For multiple loops, turn this OFF, and after surface meshing the loops, run **Edit Mesh > Repair Mesh > Flood Fill/Make Consistent** using **Make volume mesh consistent with surface mesh** followed by **Flood fill** to remove tetras in the region outside the geometry (ORFN region)

