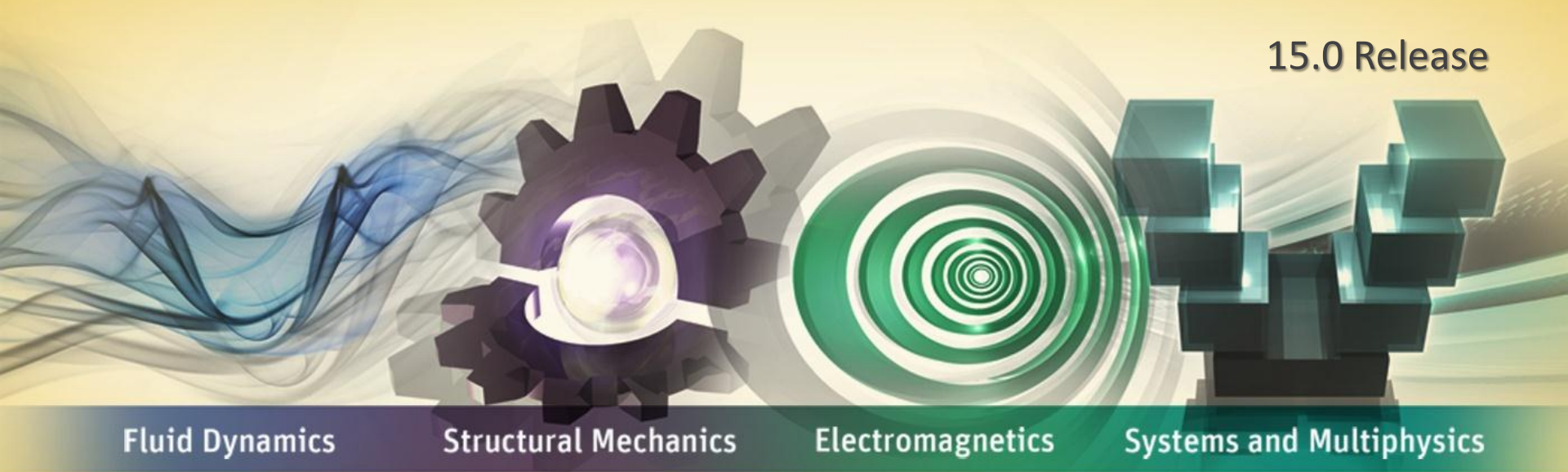


Workshop 4.1

Tetra Meshing – Engine Block

15.0 Release

A horizontal banner with a yellow 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 block structure. The text '15.0 Release' is in the top right.

Fluid Dynamics

Structural Mechanics

Electromagnetics

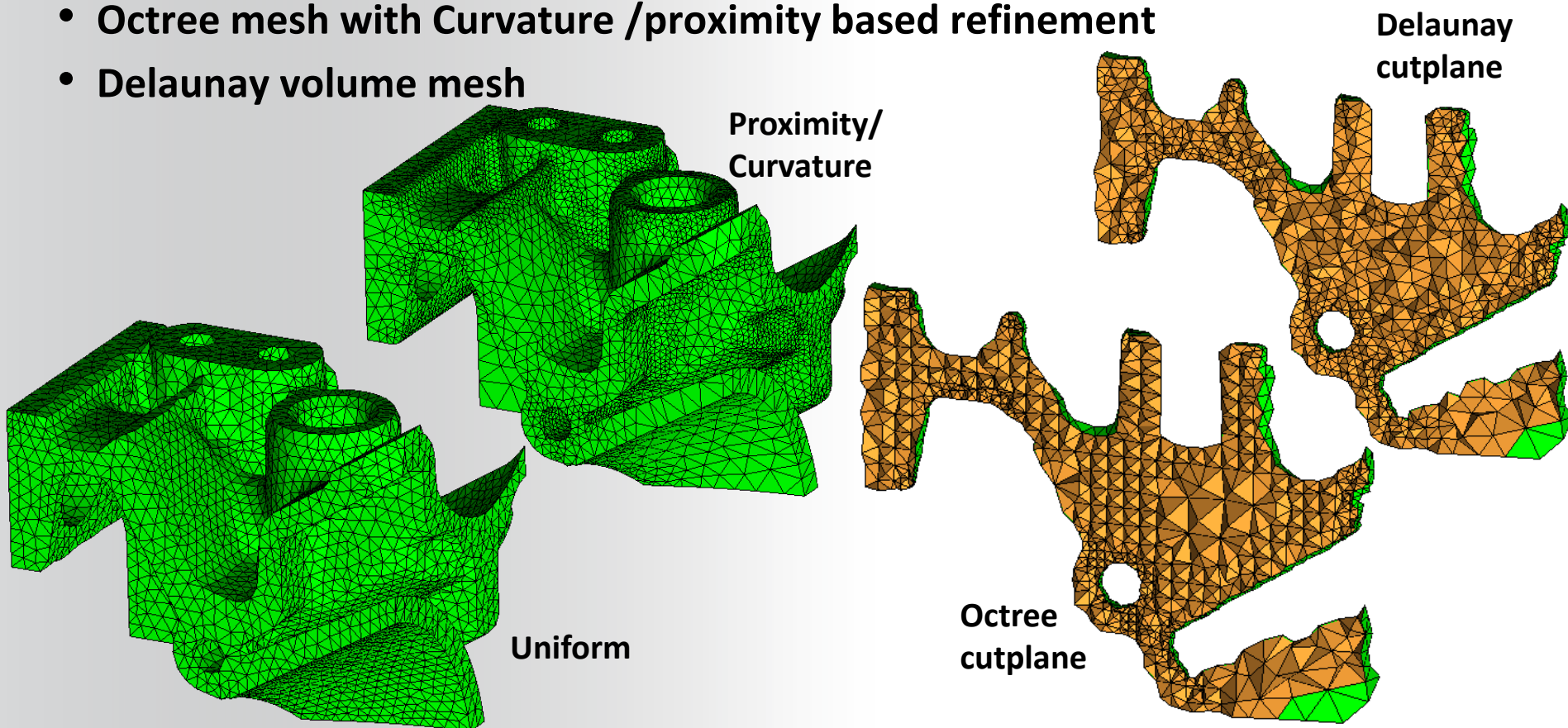
Systems and Multiphysics

Introduction to ANSYS ICEM CFD

Volume Meshing – Engine Block Meshing

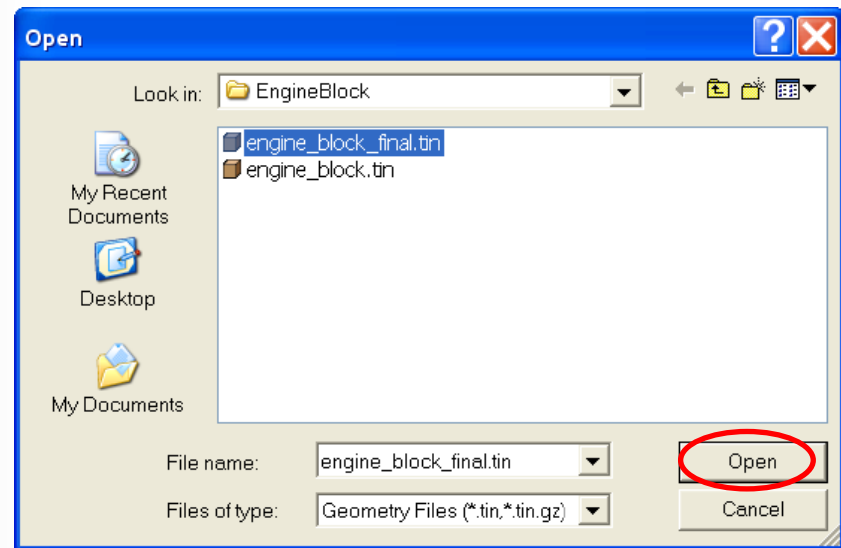
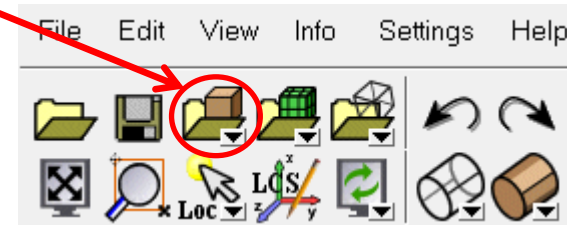
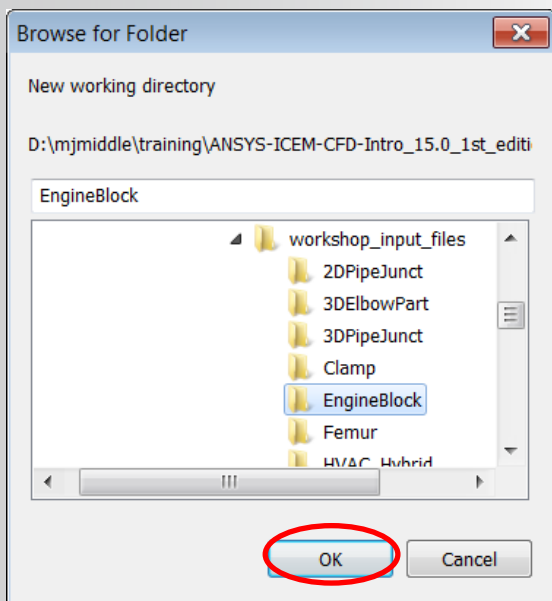
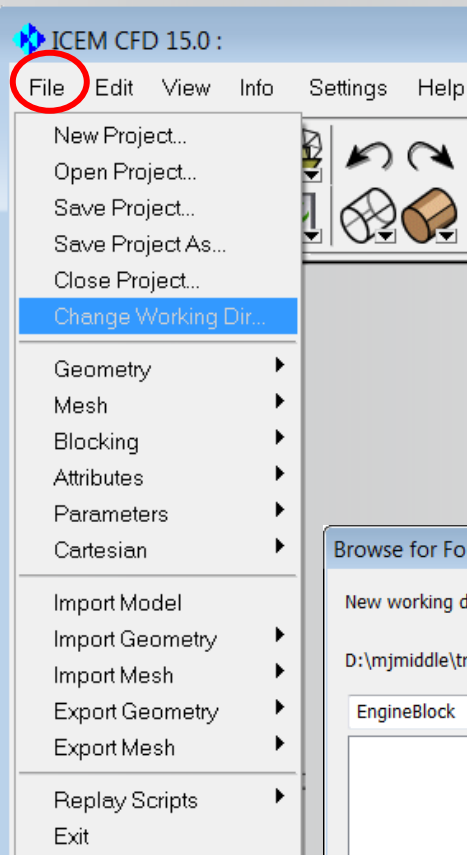
In this workshop we will create the following meshes on the engine block model,

- Octree mesh with uniform size
- Octree mesh with Curvature /proximity based refinement
- Delaunay volume mesh



Change Working Directory

- **File > Change Working Dir...**
 - Choose **EngineBlock**
 - **OK**
- **Use the quick icon to open geometry**
 - Choose **engine_block_final.tin**
 - **Open**

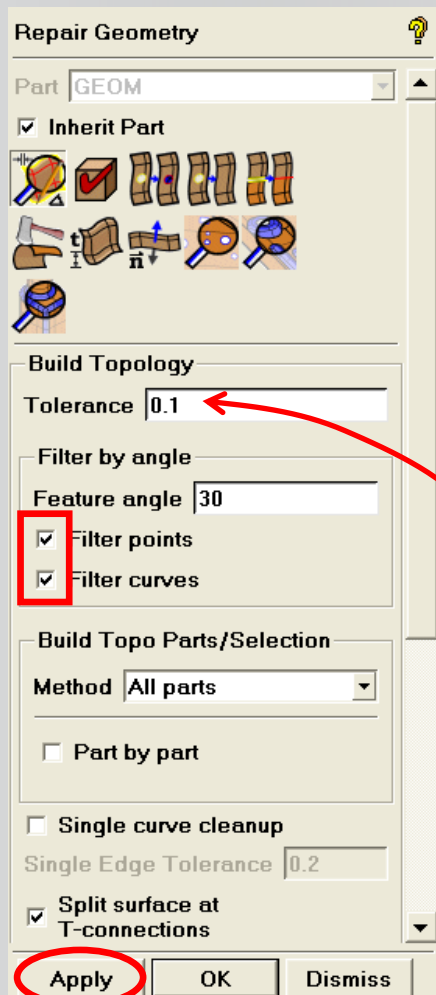


Build Topology

Geometry

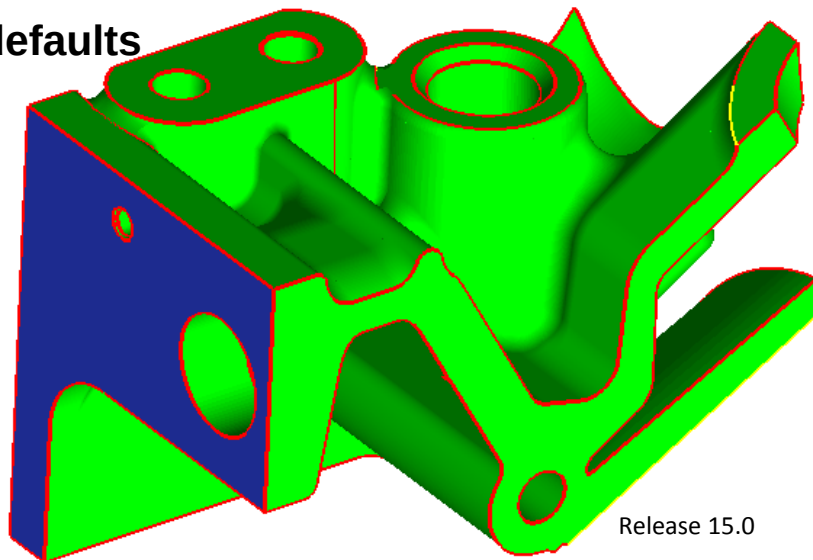


- **Geometry > Repair Geometry > Build Diagnostic Topology**



- Set **Tolerance** = 0.1
- Toggle on **Filter Points** and **Filter Curves**
 - “Filter out” or deactivate superfluous curves
 - Curves deleted where surfaces meet at a smooth (tangent) transition (< 30 degrees)
 - Points deleted where curves meet smoothly (< 30 degrees)
- Accept all other defaults
- **Apply**

Remember that the tolerance estimate is the allowable gap between surfaces



Set Mesh Sizes

Mesh



- Define volume mesh size

- Mesh > Global Mesh Setup > Global Mesh Size*

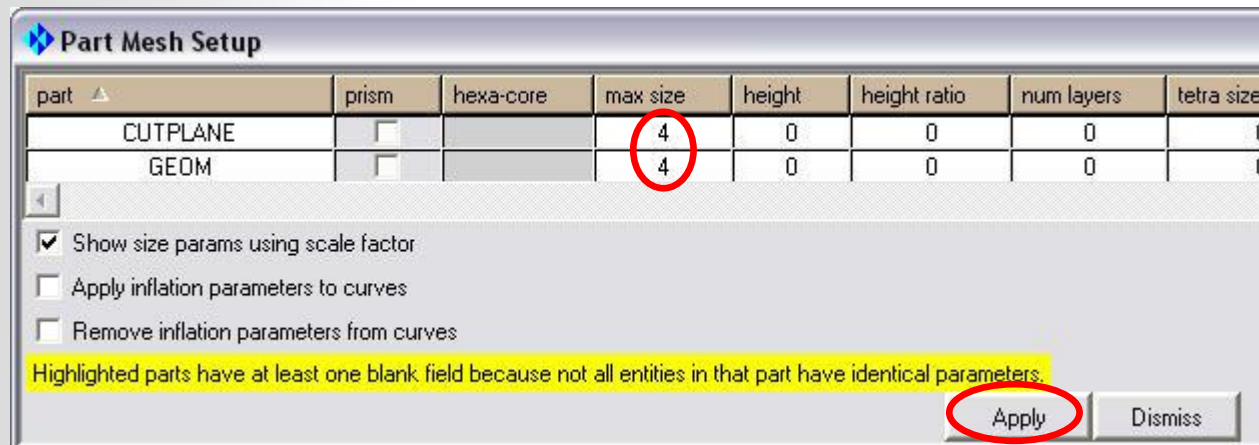
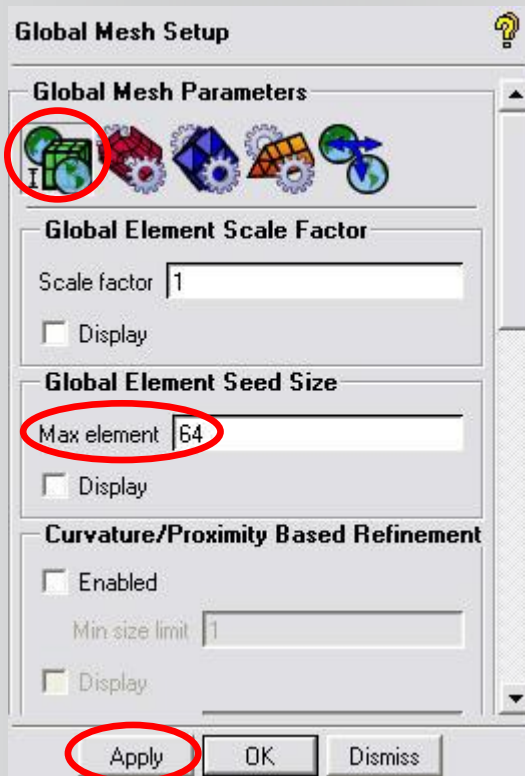
- Set **Max element** = 64
 - Relatively arbitrary, as this thin-walled part will have a volume mesh size mostly defined by the surface sizing
 - Click **Apply**

- Define surface mesh size via parts

- Mesh > Part Mesh Setup*

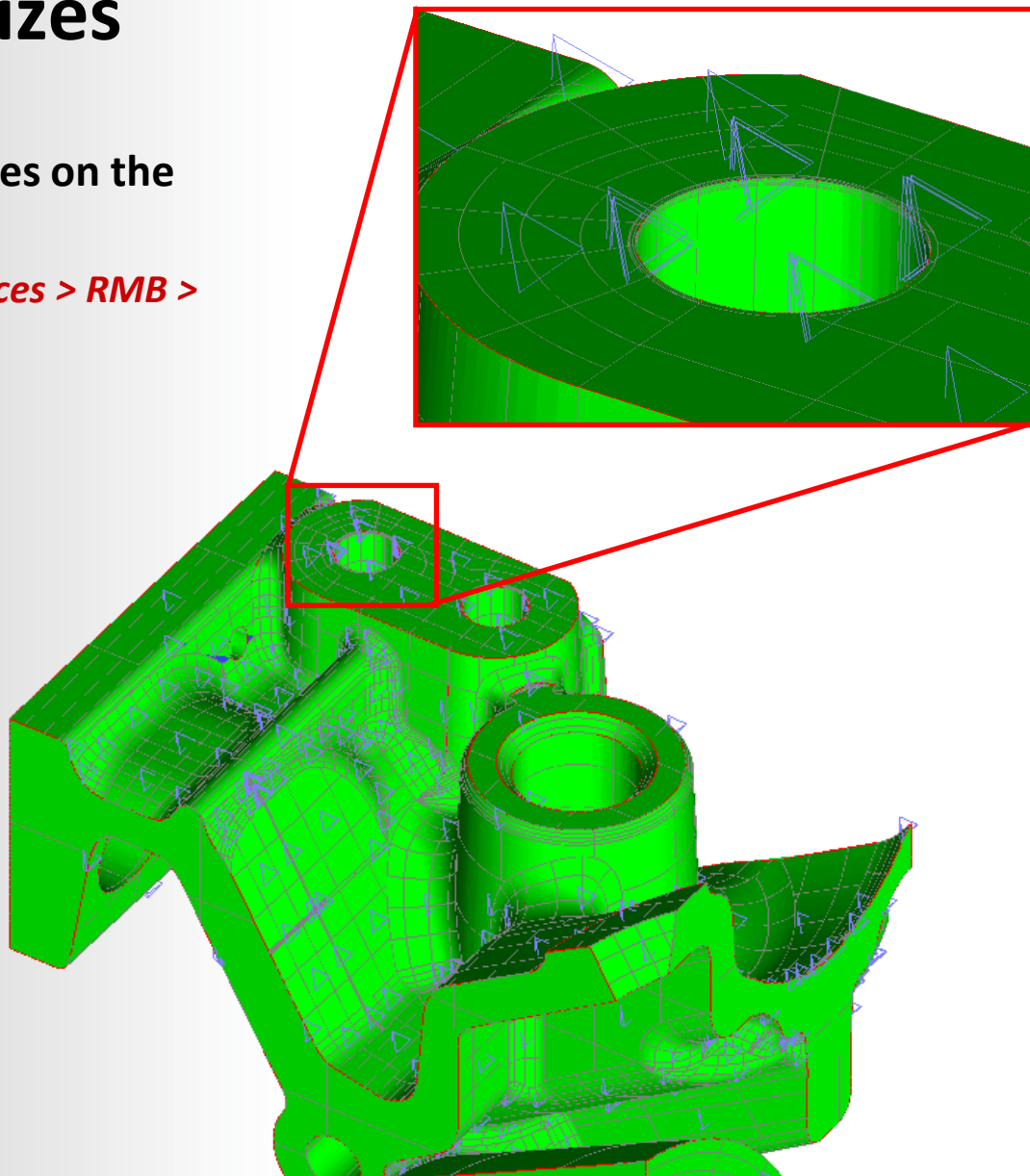
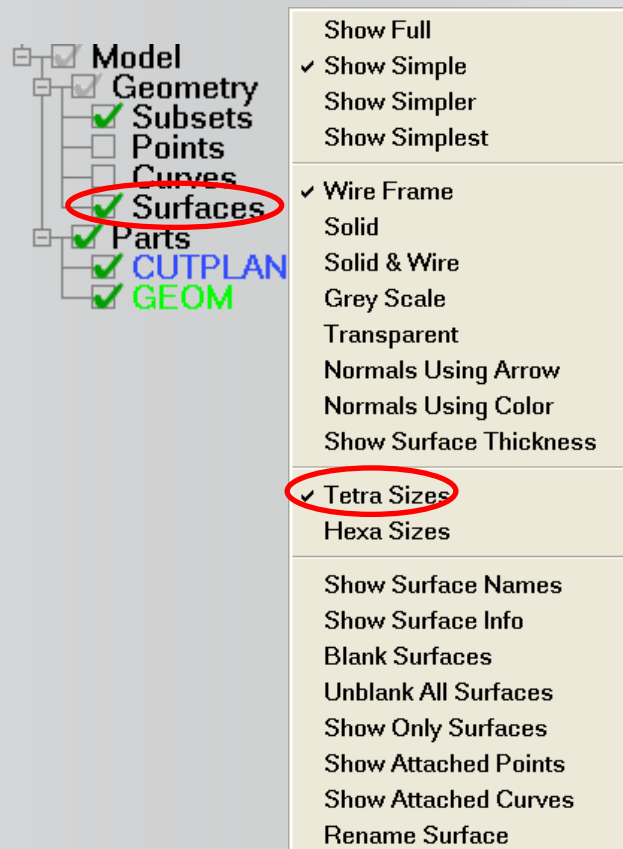
- Set **Max Size** of 4 for both **CUTPLANE** and **GEOM**
 - Click **Apply**
 - Click **Dismiss**

Mesh



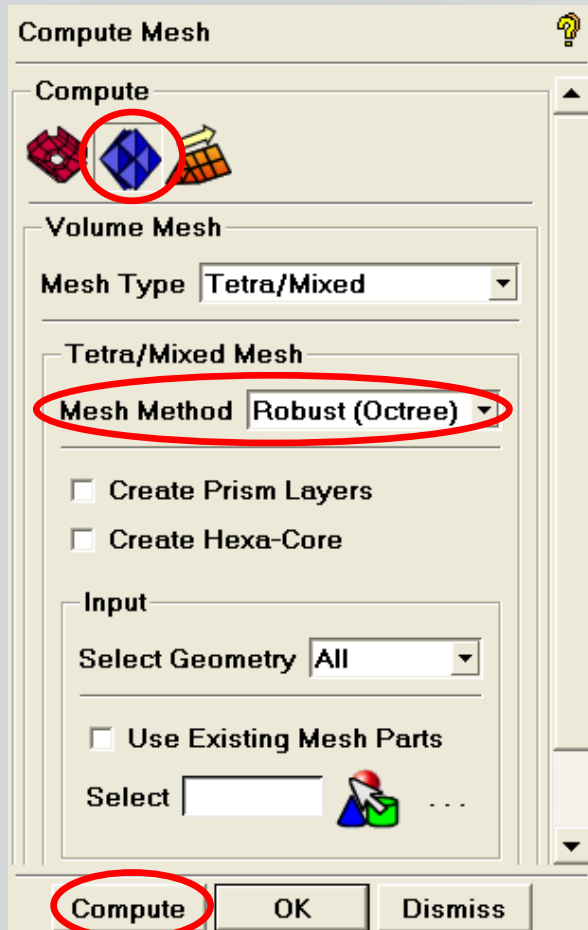
View Tetra Sizes

- Display icons representing the sizes on the surfaces
 - In model tree, *Geometry > Surfaces > RMB > Tetra Sizes*

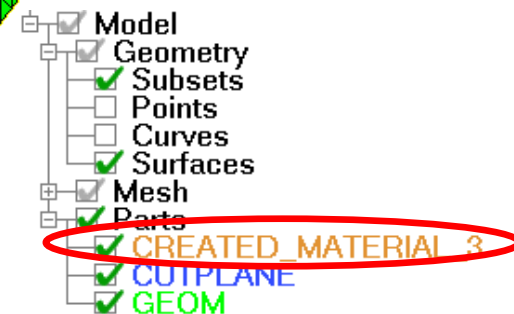
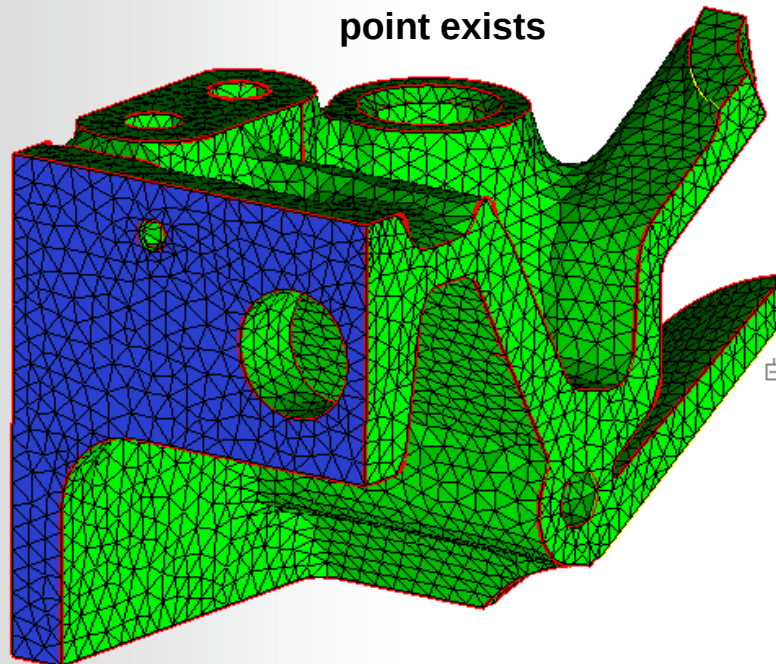


Compute Mesh

Mesh



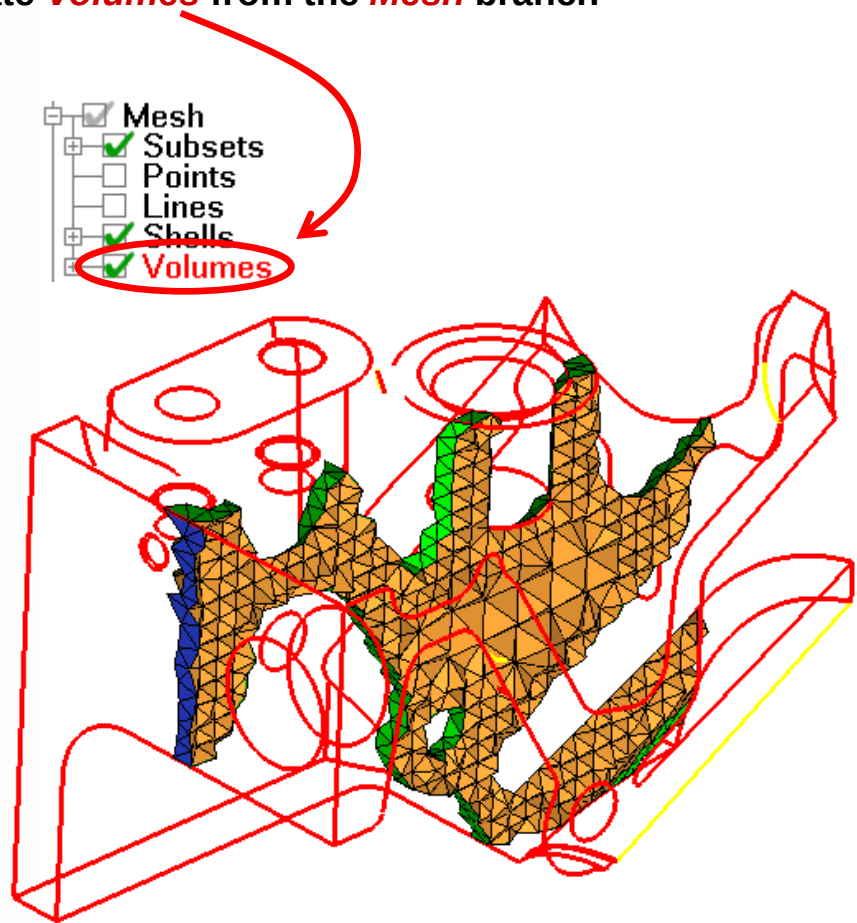
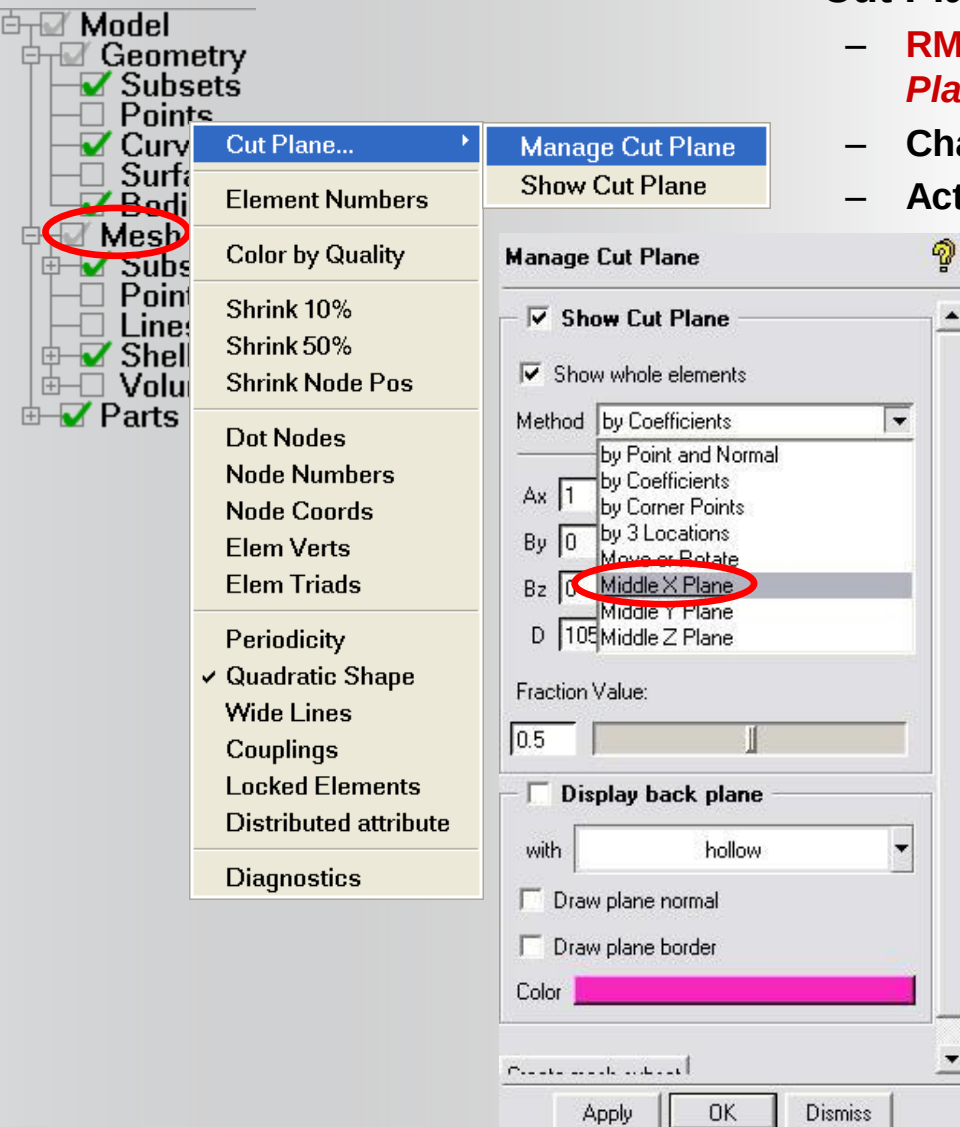
- Select **Mesh > Compute Mesh > Volume Mesh**
 - Set **Mesh Method > Robust (Octree)** (default)
 - **Compute**
 - **Note additional Part**
 - **CREATED_MATERIAL_***
 - Defines new part for volumetric tetras
 - This is the default naming when no material point exists



Cut Plane View

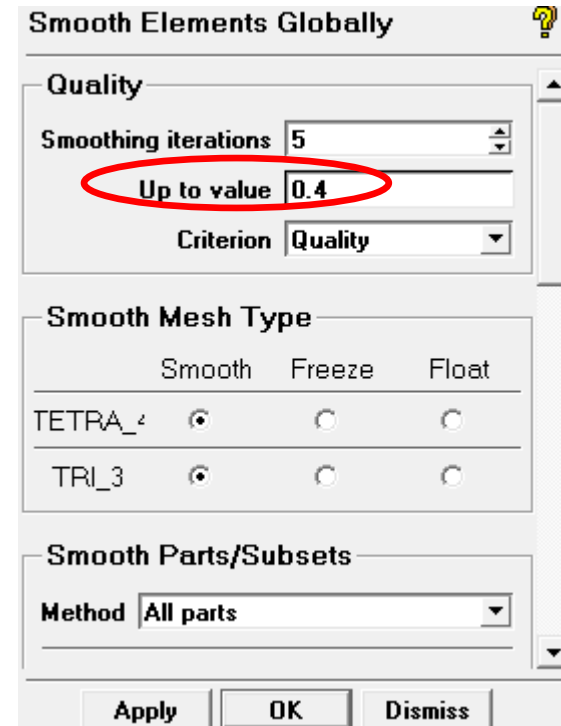
- Cut Plane View

- RMB on **Mesh** in the model tree and select **Cut Plane > Manage Cut Plane**
- Change the Cut Plane **Method** to **Middle X Plane**
- Activate **Volumes** from the **Mesh** branch

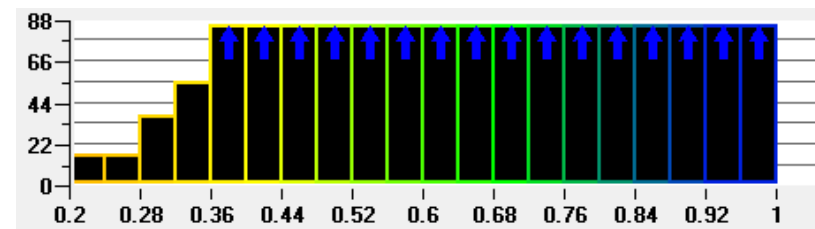




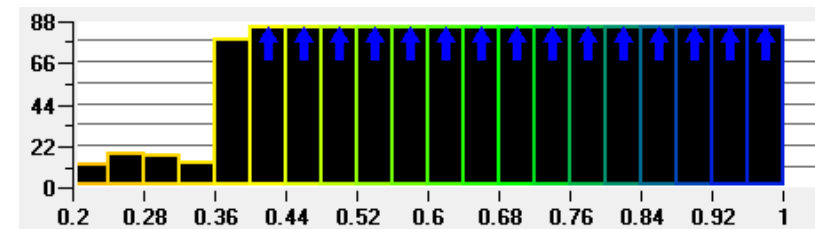
- Improve overall mesh quality
 - Select **Edit Mesh > Smooth Mesh Globally**
 - Note Histogram
 - In lower right hand corner
 - Use default for **Smoothing iterations** and **Criterion**
 - Set **Up to Value** to **0.4**
 - **Apply**
 - Note change in histogram after complete
 - You can run the smoothing function several times doing 5 iterations each time



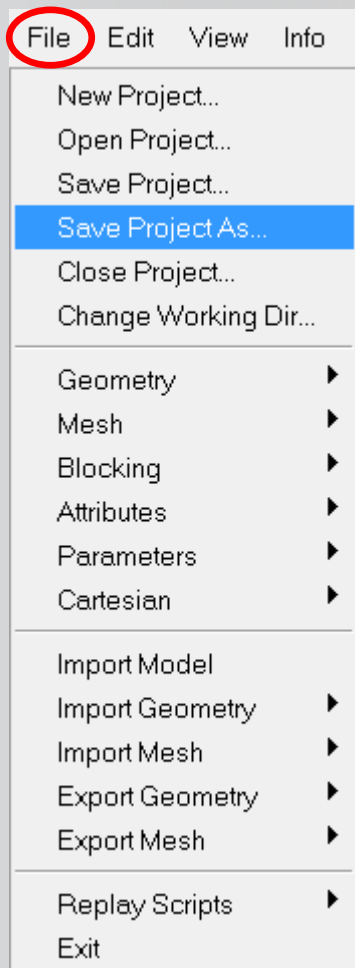
Initial quality



Final quality after smoothing

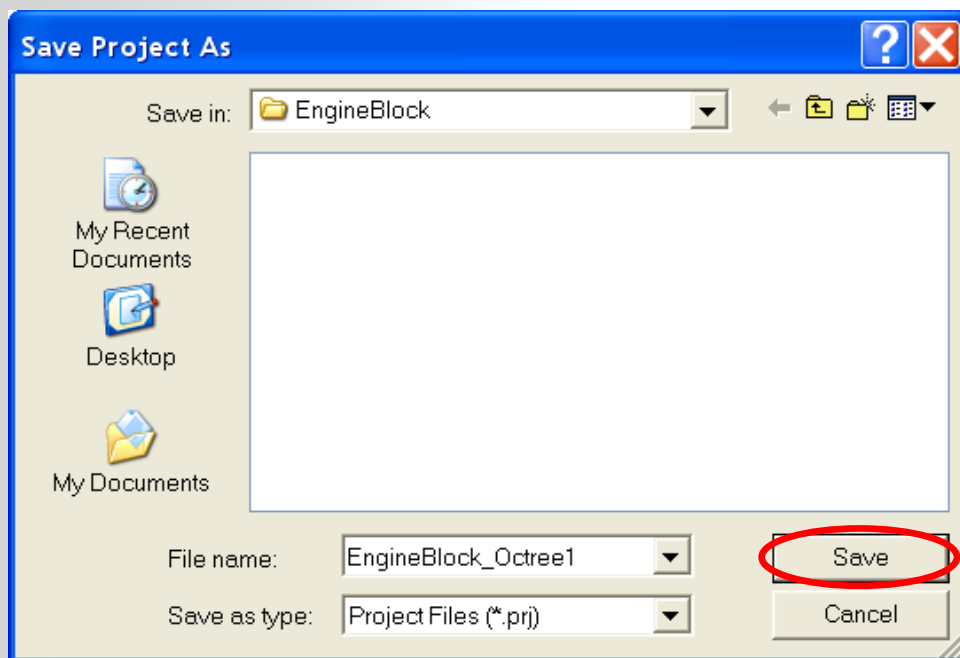


Save Project



- **Save Project As...**

- Select **File > Save Project As...**
- This will directly open the **Engineblock** directory as we have set the working directory already
- Enter project name **EngineBlock_Octree1**
- Press **Save**



Curvature/Proximity Based Refinement

Mesh



Global Mesh Setup



Global Mesh Parameters



Global Element Scale Factor

Scale factor 1

☐ Display

Global Element Seed Size

Max element 64

☐ Display

Curvature/Proximity Based Refinement

☒ Enabled

Min size limit 2

☐ Display

Elements in gap 1

Refinement 10

Apply

OK

Dismiss

- Now, let's refine the mesh in areas to better capture the details of the part. To avoid defining mesh size for every part/entity:

- Set max size for all parts/surface
- Allow global curvature/proximity sizing to define all mesh sizes for the model.

Global Mesh Parameters

- *Mesh > Global Mesh Setup > Global Mesh Size*
- Set *Max Element* = 64
- Set *Curvature/Proximity Based Refinement* to *Enabled*
- Set *Min Size Limit* to 2
- *Apply*

Part Mesh Parameters

- *Mesh > Part Mesh Setup*
- Set *Max Size* of 8 for both CUTPLANE and GEOM
- Click *Apply*
- Click *Dismiss*

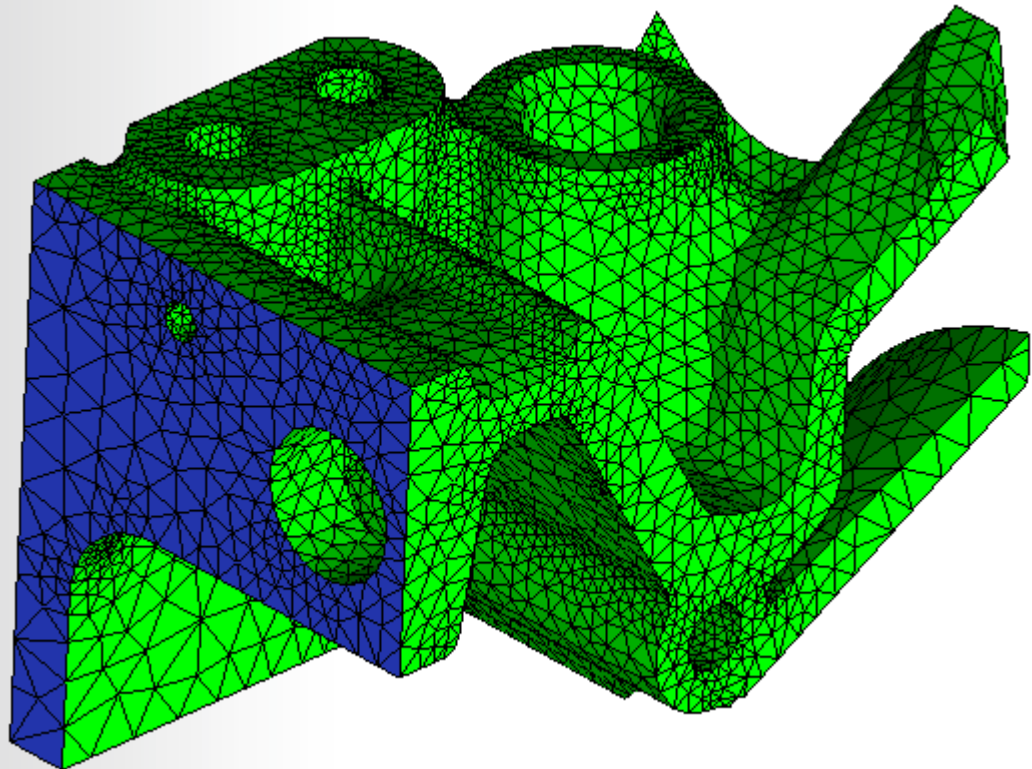
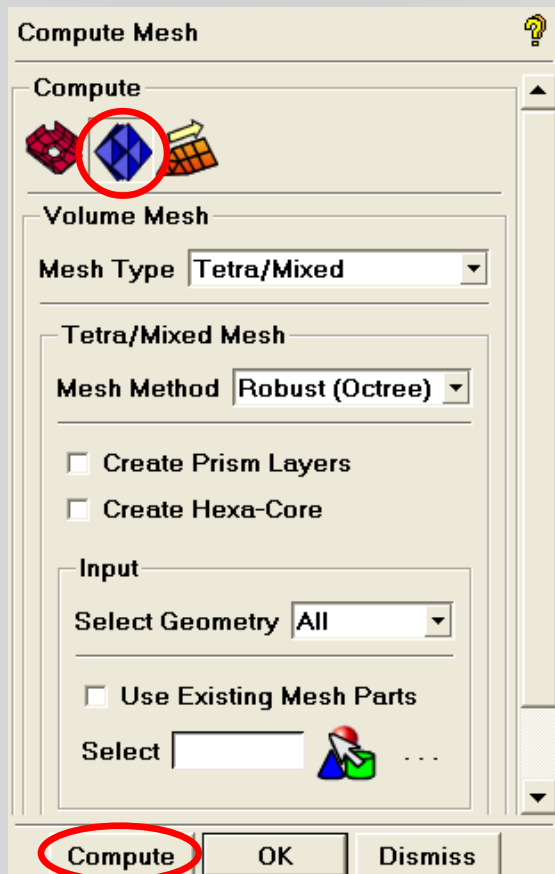
Mesh



Mesh



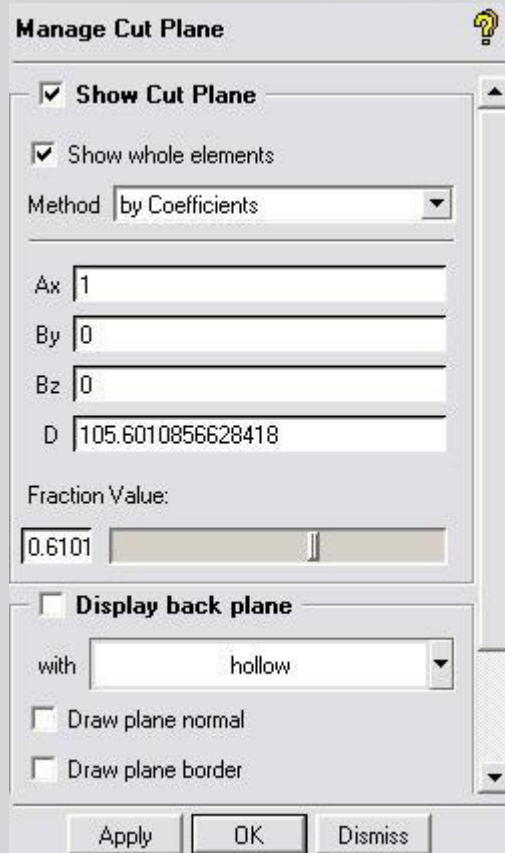
- Close the existing mesh before creating the new mesh using **File > Mesh > Close Mesh**
- **Mesh > Compute Mesh > Volume Mesh**
 - **Compute**
 - Note refinement near small radii of curvature fillets



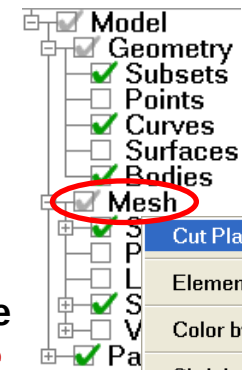
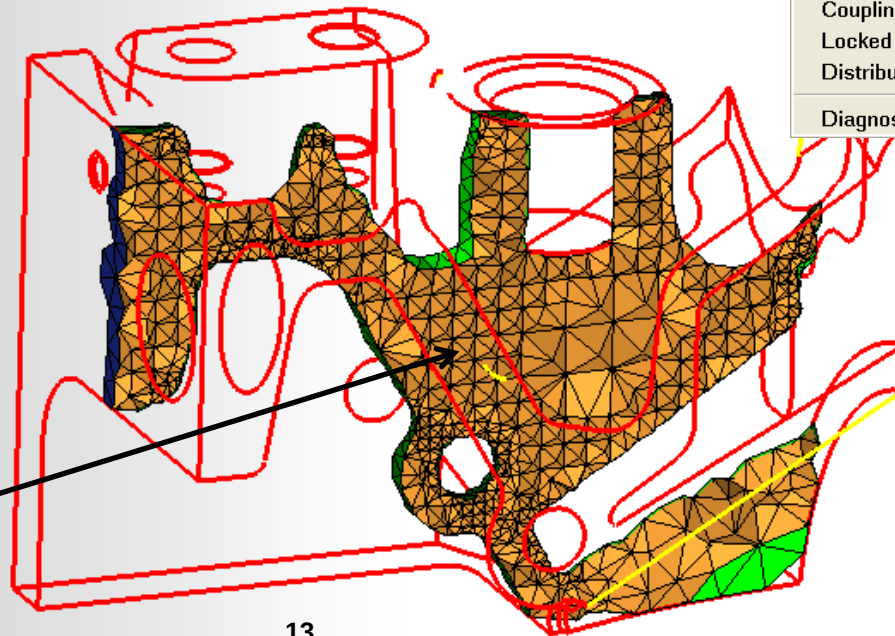
Cut Plane View

• Cut Plane View

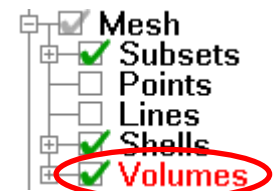
- RMB on **Mesh** in the model tree and select **Cut Plane > Manage Cut Plane**
- Change the Cut Plane **Method** to **Middle X Plane**
- Activate **Volumes** from the **Mesh** branch



Note the square looking tetras that are characteristic of the octree method



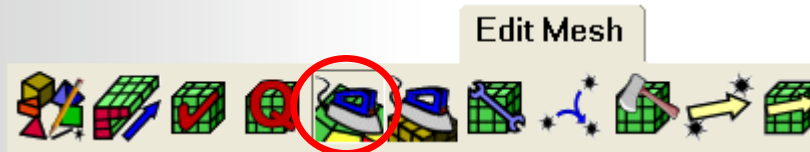
Manage Cut Plane
Show Cut Plane



Investigate Mesh Quality

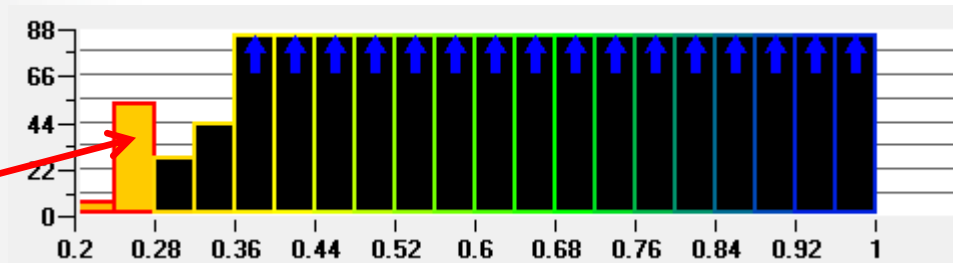
- **Smooth** mesh as before

- **Up to value** of 0.4
- Note still some elements below a Quality of 0.3 (~ 0.28)



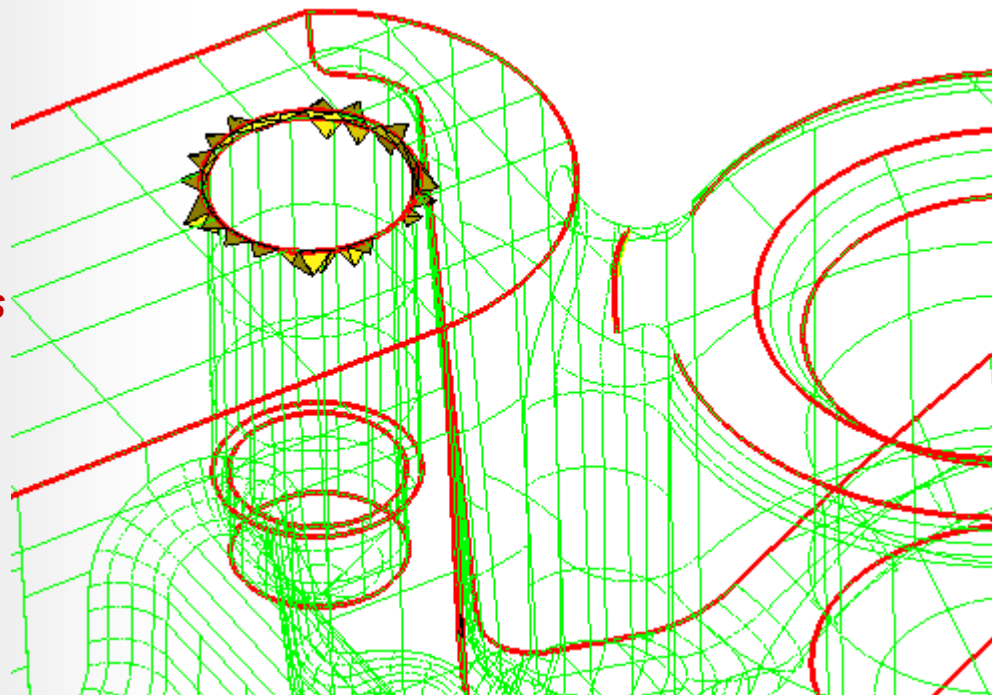
- Highlight these elements

- Click on the lowest bar in the Histogram



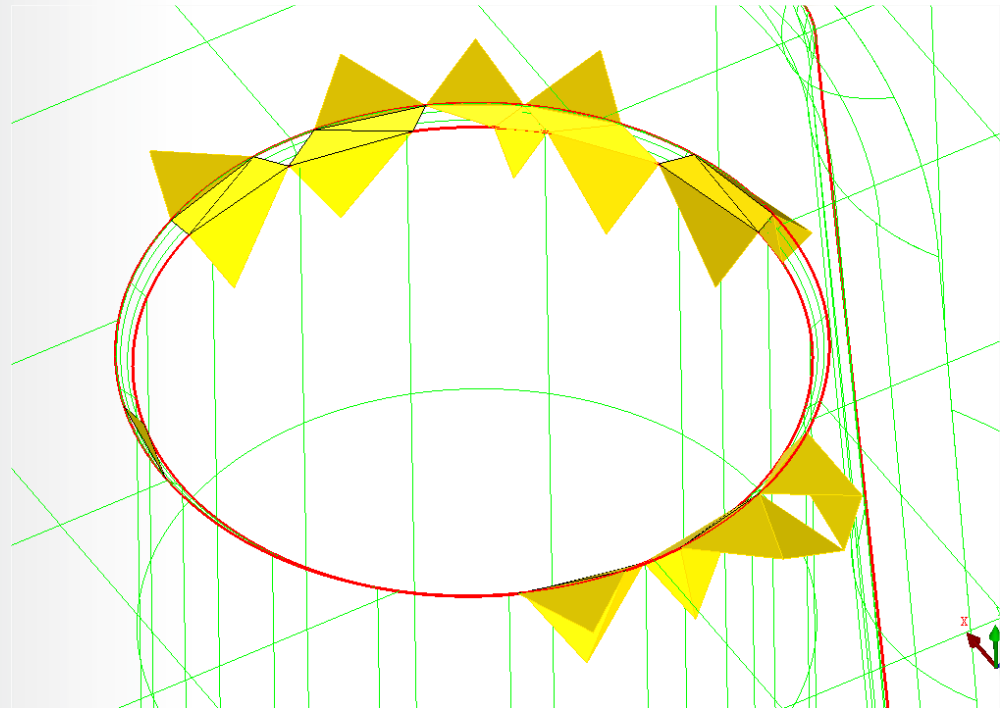
- Find the highlighted elements in the model

- **RMB** on **Mesh** > **Cut plane...** > **Show Cut Plane** to turn off the cut plane
- Uncheck the **Shells** and **volumes** in the model tree
- Elements highlighted in yellow appear near one of the holes



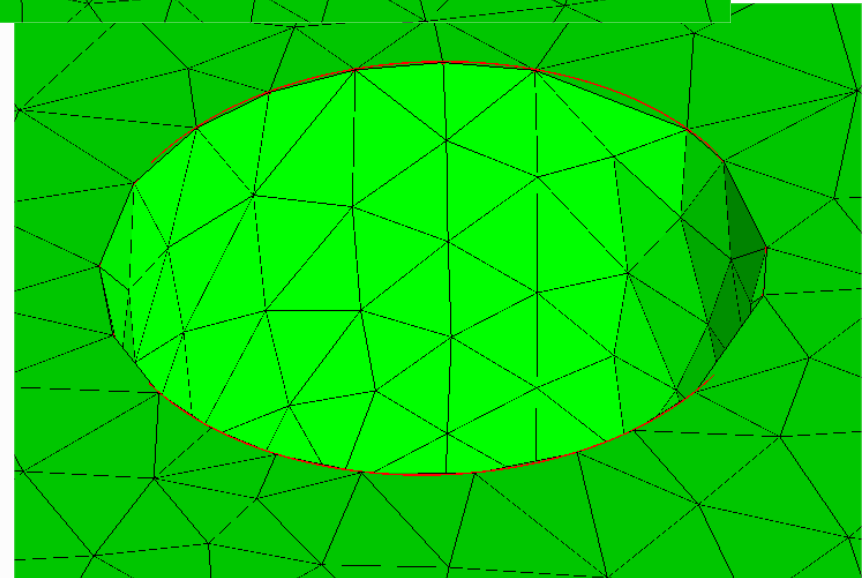
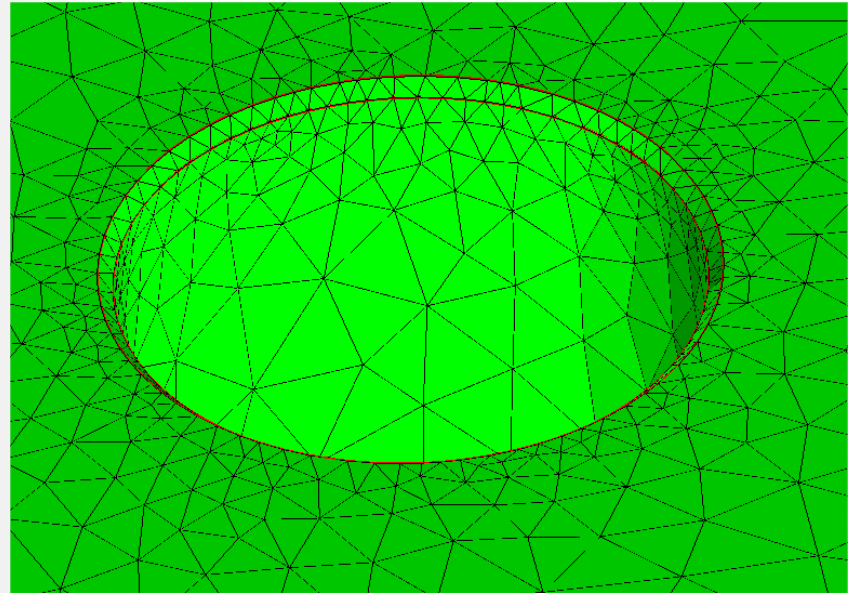
Investigate Mesh Quality

- Why poor element quality here?
 - Two curves present to define the chamfer on the top of the hole
 - Gap between the two curves is more than 5 times smaller (0.35) than our defined minimum element size (2)
 - Large TRI and TETRA elements are then squished into this space
- How to fix this area?
 - Set a smaller mesh size on the chamfer surfaces to make sure this area is resolved
 - ➡ Increased element count
 - Remove the lower curve to remove constraint on mesher
 - ➡ Lack of conformance with geometry



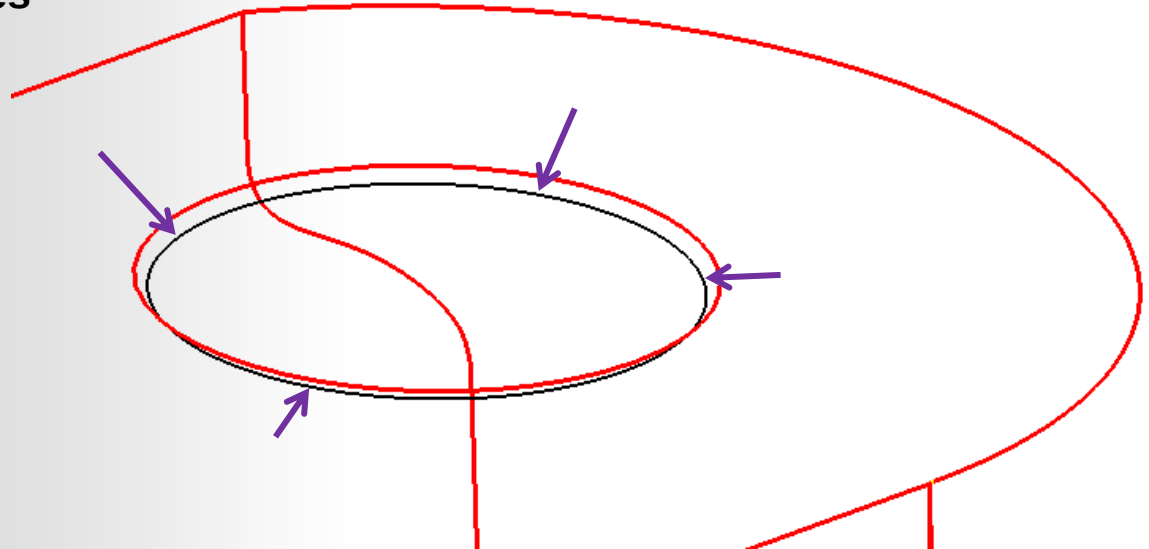
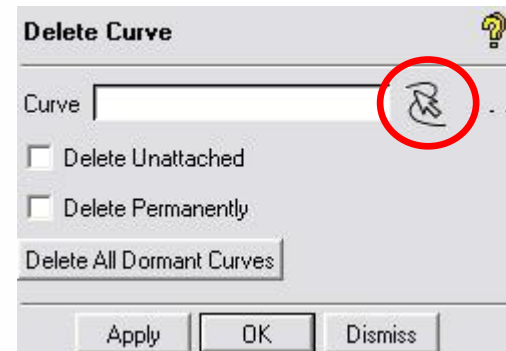
Investigate Mesh Quality

- Mesh shown with a mesh size of 0.5 defined on the various chamfer surfaces
 - Chamfer surfaces now resolved with reasonably sized elements
- Mesh shown after removing the lower curve from the model
 - Surface mesh is now allowed to walk over this chamfer feature, formally adhering only to the upper edge
 - Did not increase element size



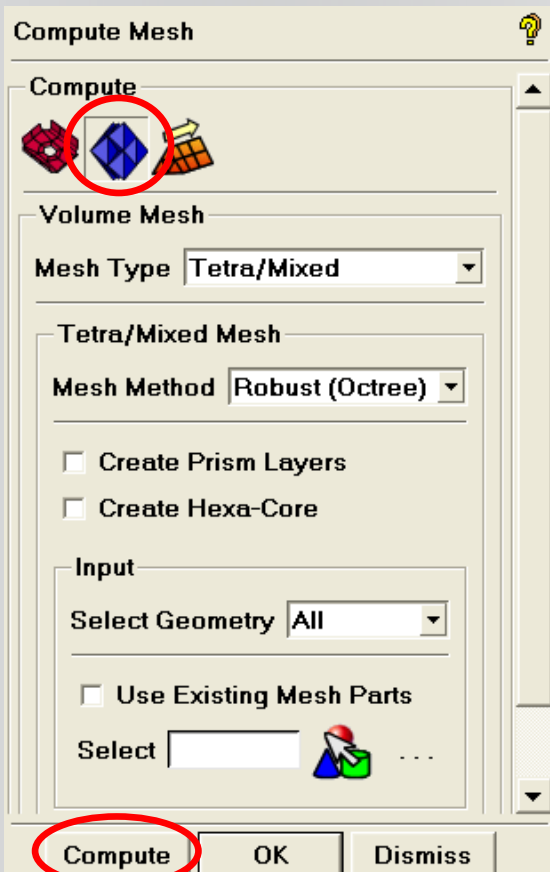
Adjust Geometry and Remesh

- In this case, let's assume that this chamfer definition is not important for our analysis
- Close this mesh without saving
 - ***File > Mesh > Close Mesh***
- We will delete these curves at the chamfered hole location to obtain a better mesh in this zone
- ***Geometry > Delete Curve***
 - Select the 4 lower curves shown for deletion

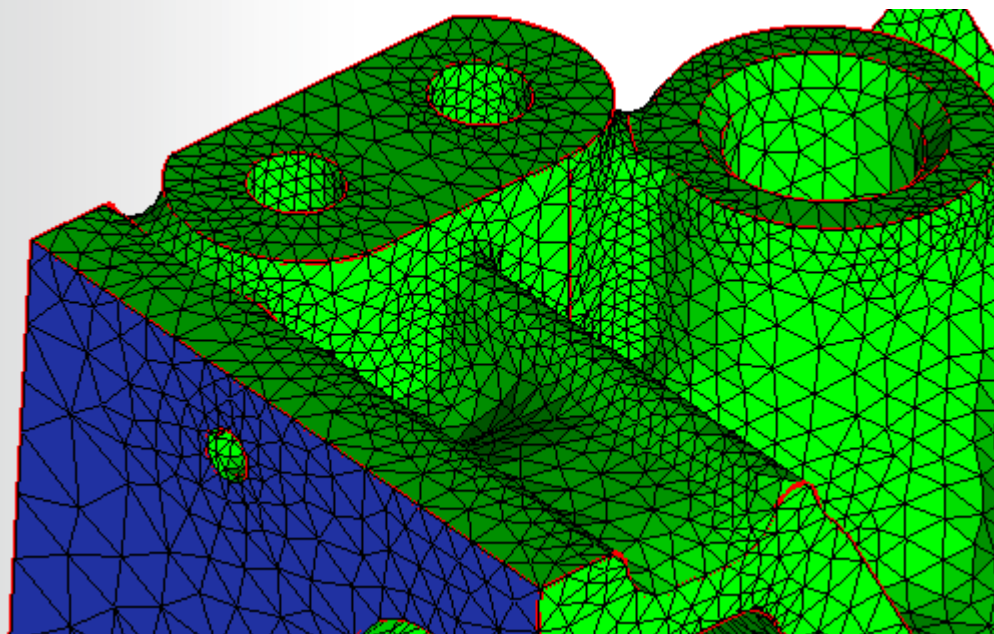


Remesh and Smooth

Mesh

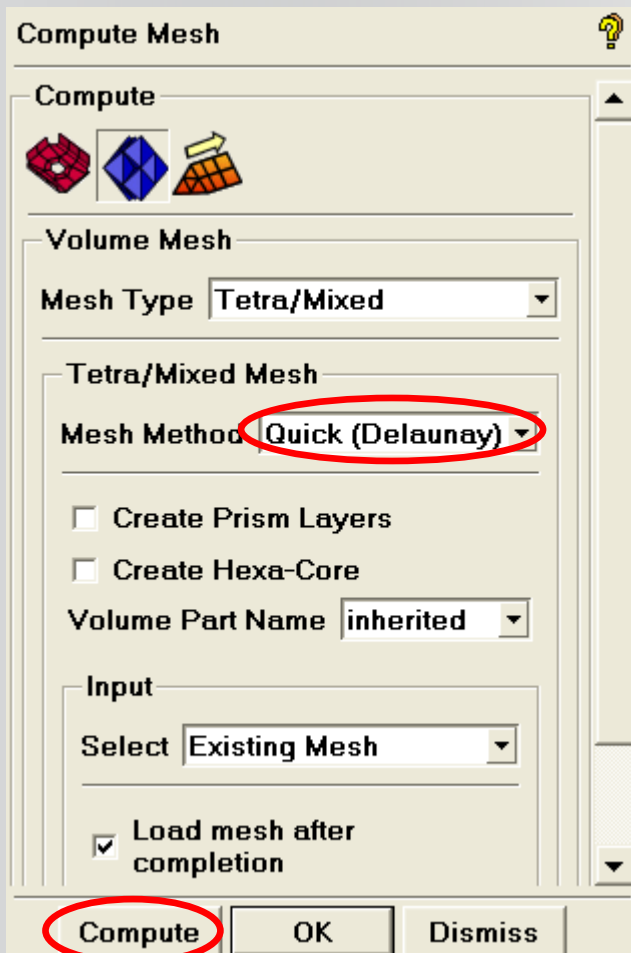


- Re-mesh and Smooth
 - *Mesh > Compute Mesh > Volume Mesh*
 - *Compute*
 - *Edit Mesh > Smooth Mesh Globally*
 - *Apply*
- *File > Save Project As..*
 - Save it as *EngineBlock_Octree2.prj*



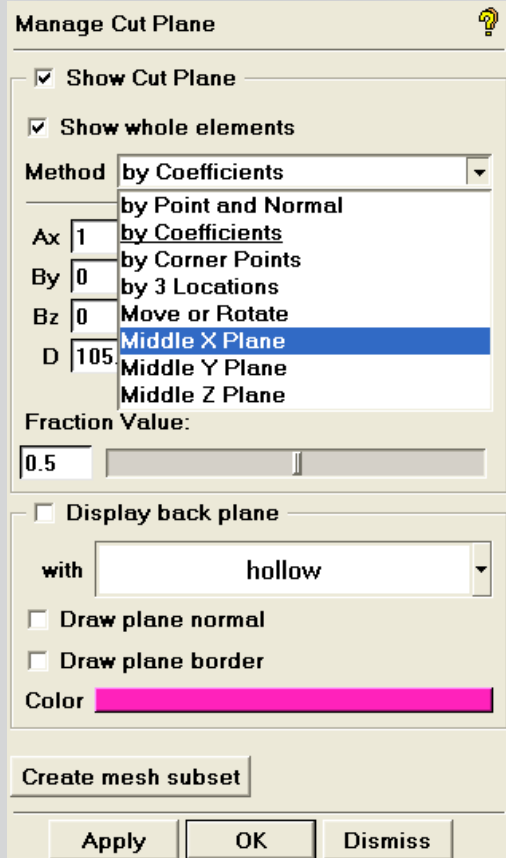
Delaunay Mesh Method

Mesh



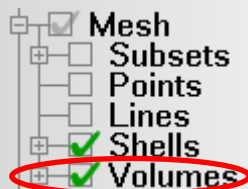
- Delaunay tetra mesh is created from existing shell mesh. If you start from geometry, ICEM CFD will create the shell mesh first with the patch dependent surface mesher using the global and local shell mesh settings and then fill the volume with Delaunay tetra.
- We can leverage the robustness advantage of the Octree mesher to create the robust surface mesh and combine it with the Delaunay mesher for a smoother element size transition in the volume
- Leave the current Octree mesh loaded
- Go to **Mesh > Compute Mesh**
 - Change the **Mesh Method** to **Quick (Delaunay)**
 - The **Input** field is automatically set to **Existing Mesh**
- **Compute Mesh**
- ICEM CFD throws away the Octree volume mesh, but keeps the surface mesh to seed the Delaunay volume mesher

Cutplane View



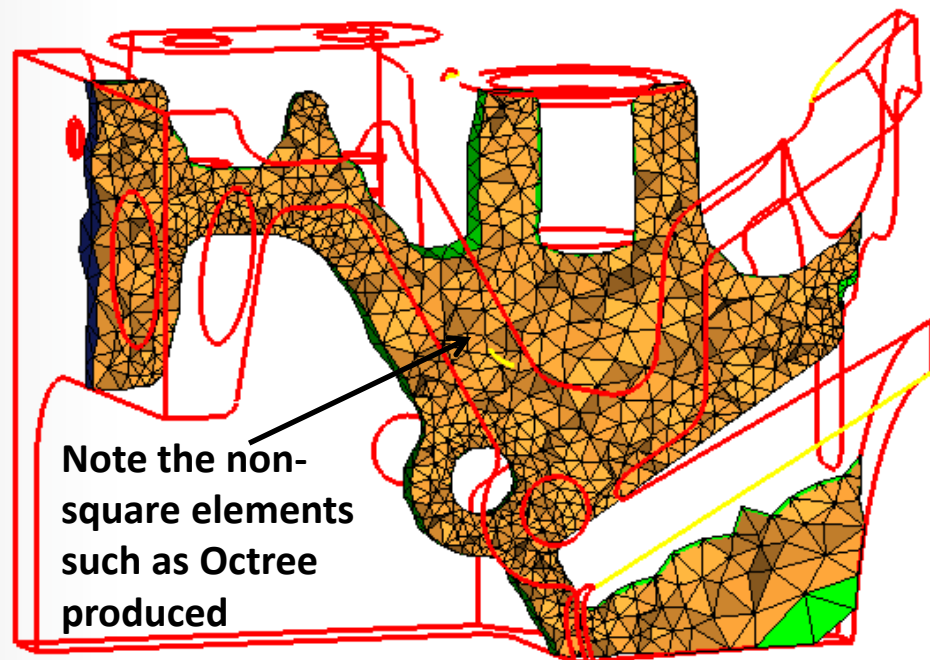
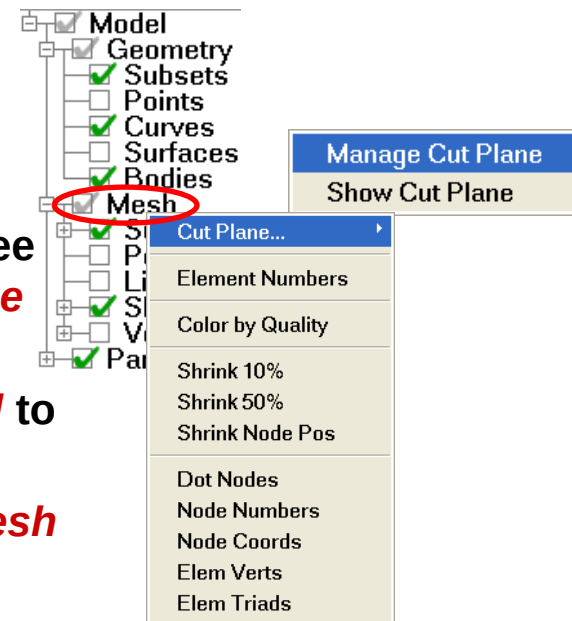
• Cut Plane View

- **RMB** on **Mesh** in the model tree and select **Cut Plane > Manage Cut Plane**
- Change the Cut Plane **Method** to **Middle X Plane**
- Activate **Volumes** from the **Mesh** branch



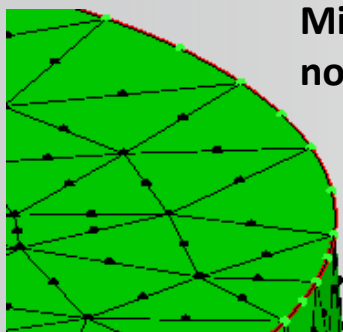
• Smooth

- **File > Save Project As...**
 - Enter **EngineBlock_delaunay.prj**

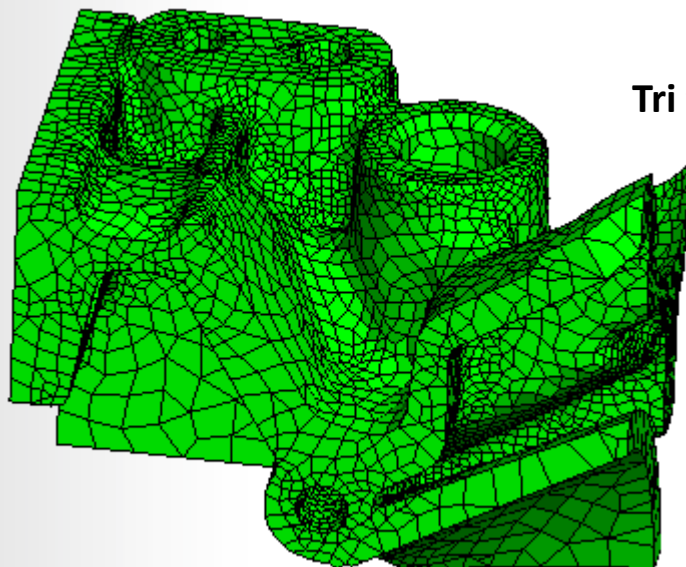


Further Investigation

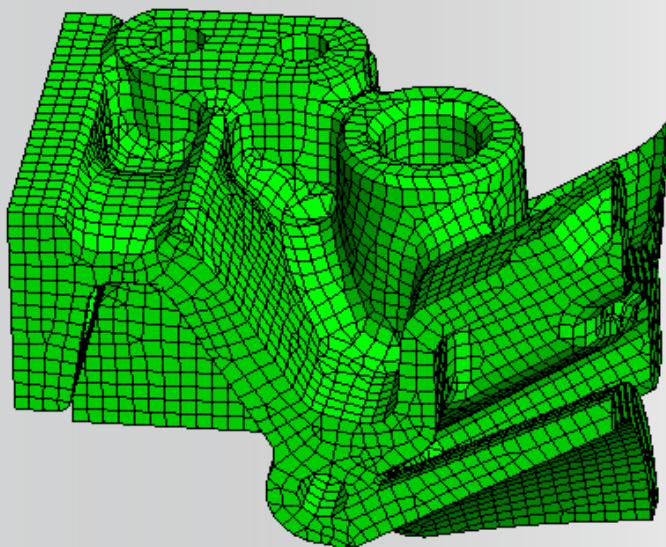
- For those that have the time, the following pages introduce additional concepts



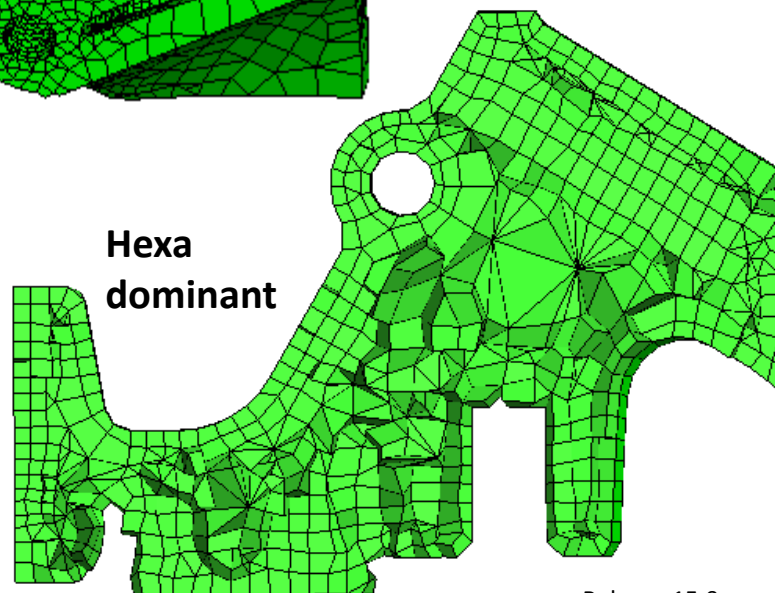
Mid side
nodes



Tri to quad



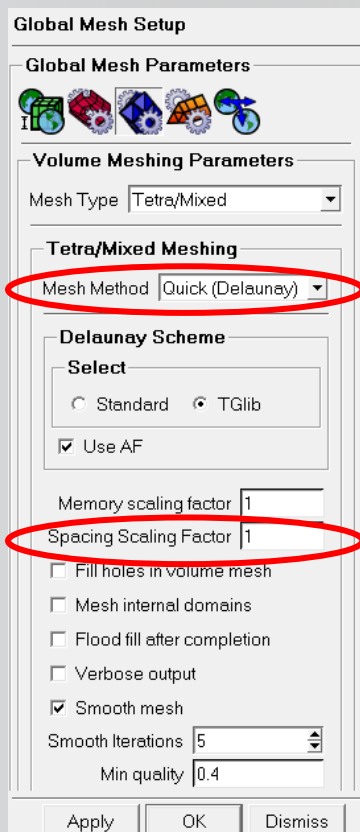
Patch
dependent
surface
mesher



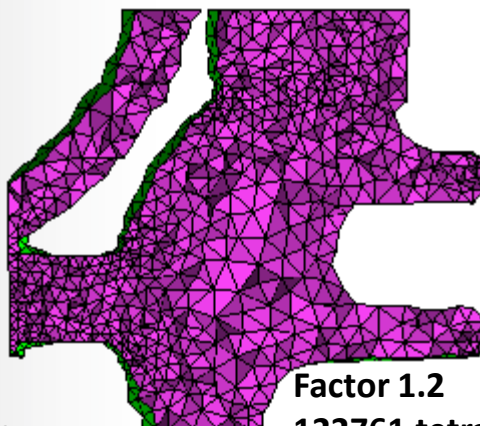
Hexa
dominant

Delaunay Growth Ratio

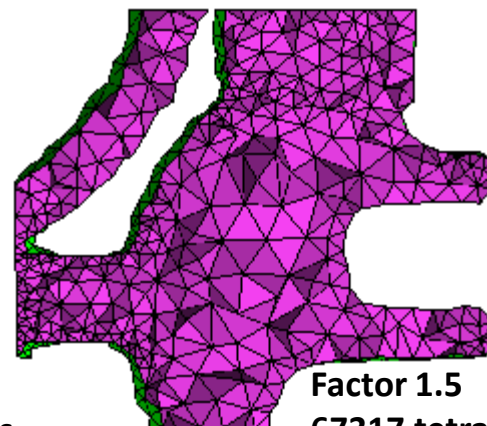
- When you meshed with Delaunay, you simply chose the Delaunay method in the **Compute Mesh** panel and accepted all default global settings.
- **Mesh > Global Mesh setup > Volume Meshing Parameters**
 - Set **Mesh Method** to **Quick (Delaunay)**
 - One of the most important parameters here is the **Spacing Scaling Factor**
 - This is the growth larger into the volume up to the **Global Element Seed Size** if it can be achieved in the space
 - The default value of 1 actually uses 1.2
 - Experiment with different values, refilling with Delaunay each time
 - Use cut plane to view internal mesh



Factor 1.05
325667 tetras



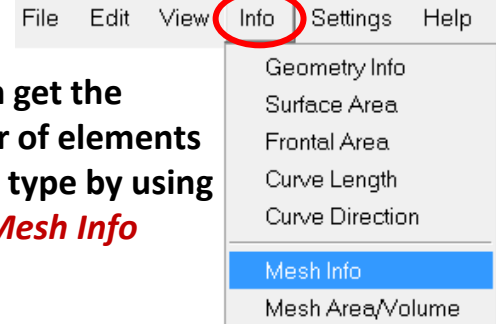
Factor 1.2
122761 tetras



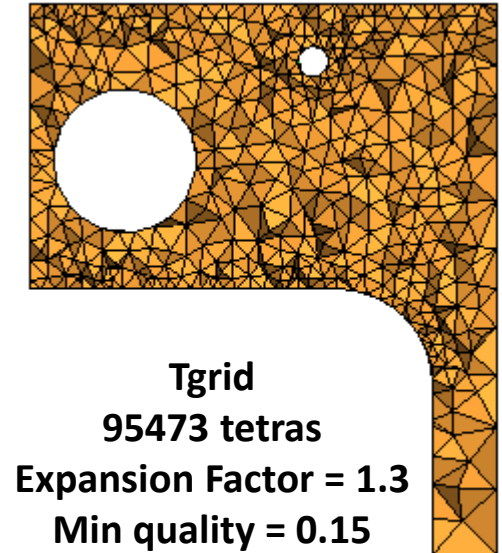
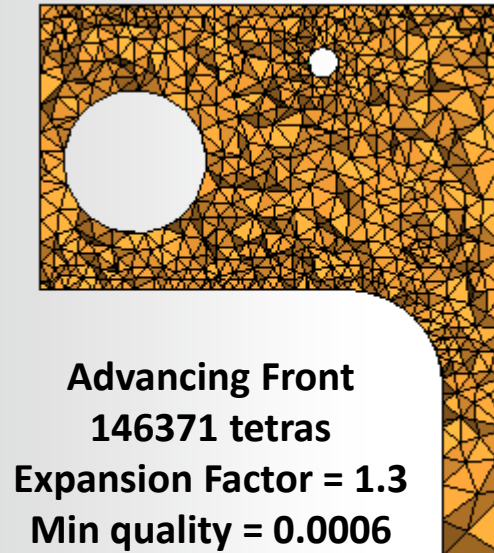
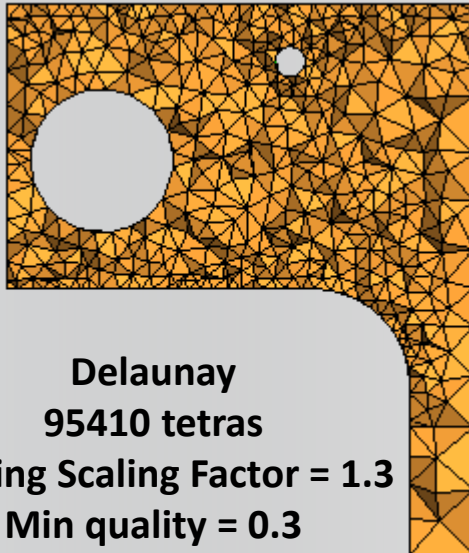
Factor 1.5
67317 tetras
Release 15.0

Compare Meshers

- Compare the 3 different tetra filling methods
 - Delaunay
 - Advancing Front
 - Tgrid
- Make sure to set the same growth ratio in the *global mesh setup* for each method in order to compare
 - For Delaunay, it is called *Spacing Scaling Factor*
 - For *Advancing Front* and *Tgrid*, it is called *Expansion Factor*
- Compare by viewing cutplanes, minimum histogram quality, and number of tetras

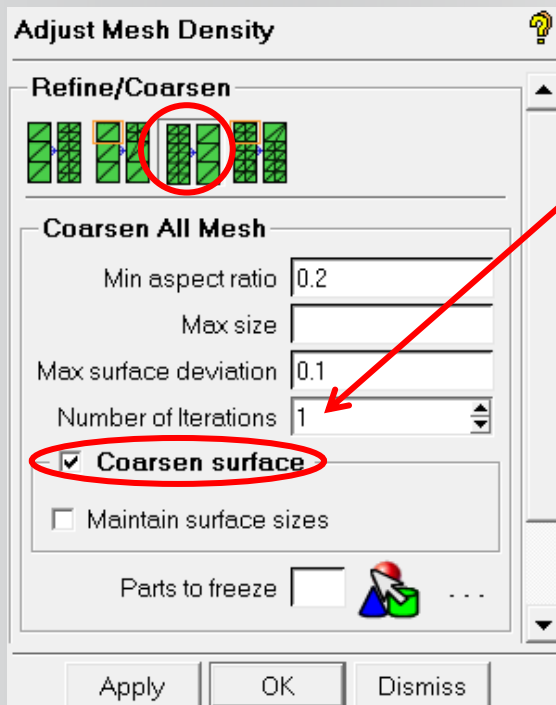


You can get the number of elements of each type by using *Info > Mesh Info*



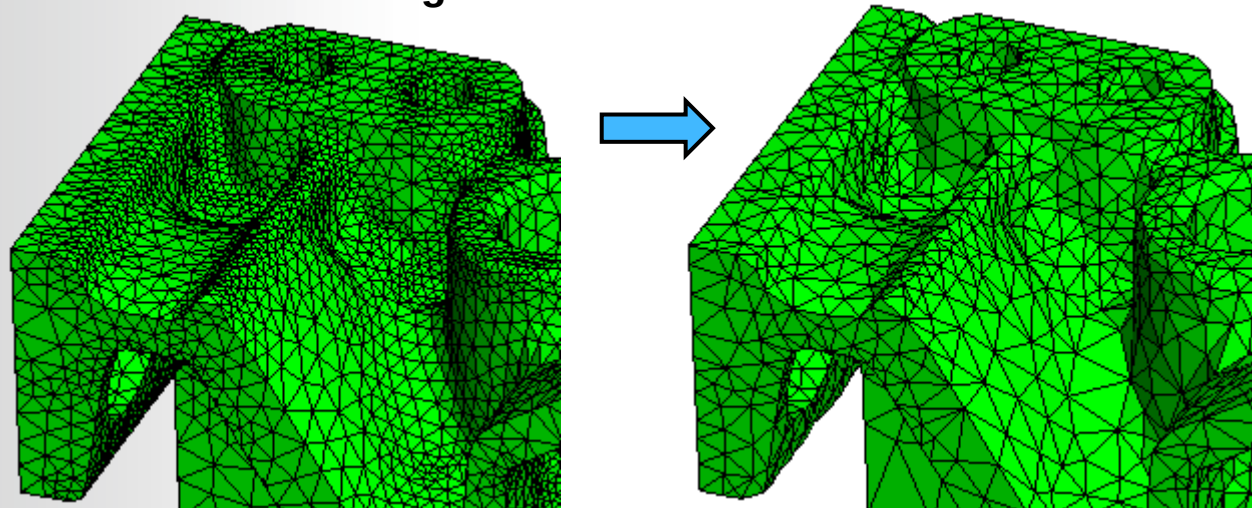
Coarsen Mesh

- If you need less elements, it can be sometimes quicker to mesh with a finer size to model features correctly, and then coarsen, rather than trying to mesh with the larger sizes from the beginning
- Edit Mesh > Adjust Mesh Density > Coarsen All Mesh**



- Check ON **Coarsen surface** or else it will only affect the volume
- Apply** with the rest of the defaults

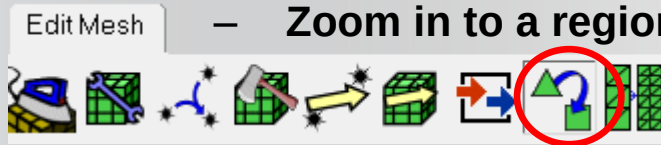
- No elements will be coarsened so as to produce a lower aspect ratio than the specified **Min aspect ratio** (0.2)
- 10 smoothing iterations are automatically done after each coarsening iteration



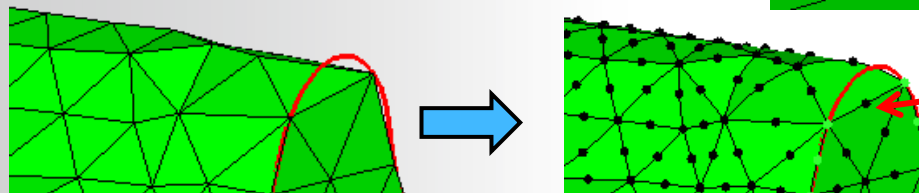
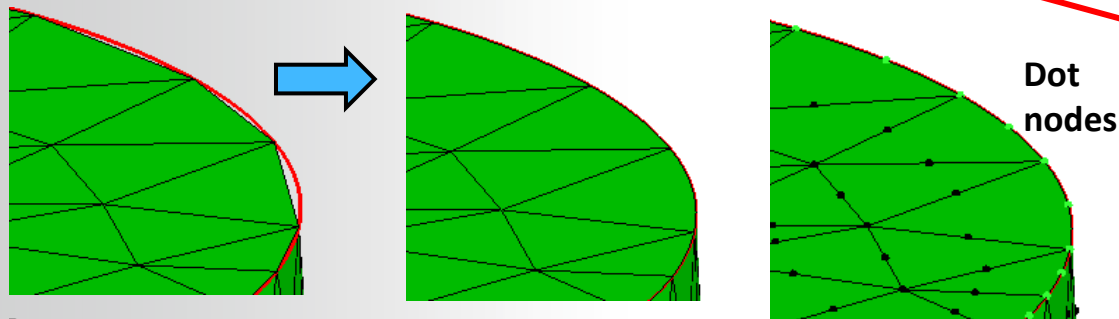
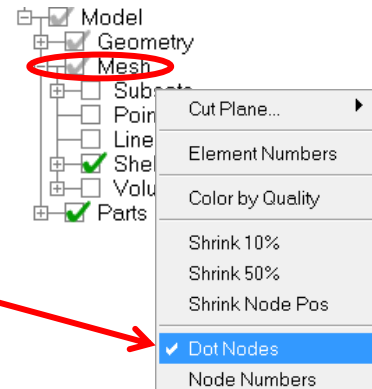
Convert to Quadratic Elements

- Load the previously saved, uncoarsened mesh before proceeding
- For mechanical analysis, linear tetras act too stiff
- Therefore, it is preferred to convert linear tetras to quadratic tetras, which means adding mid-side nodes
- **Edit Mesh > Convert Mesh Type > Create Mid Side Nodes**

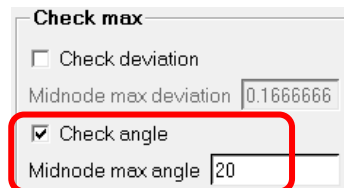
– Zoom in to a region with coarse mesh and curves to see the difference



- Press **Apply** with defaults, then **Undo**, and experiment with the quality controls (**Check deviation, angle, and chord**) and **Tolerance** limit
- Turn on **Dot Nodes** to see the mid-side nodes



This node is not projected with **Check angle** turned ON and set below 23°

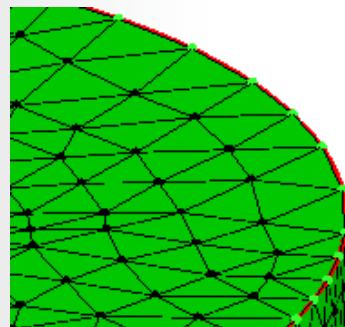
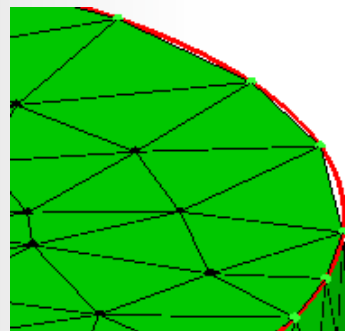
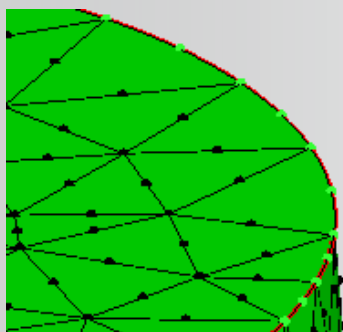


Delete Mid Side Nodes

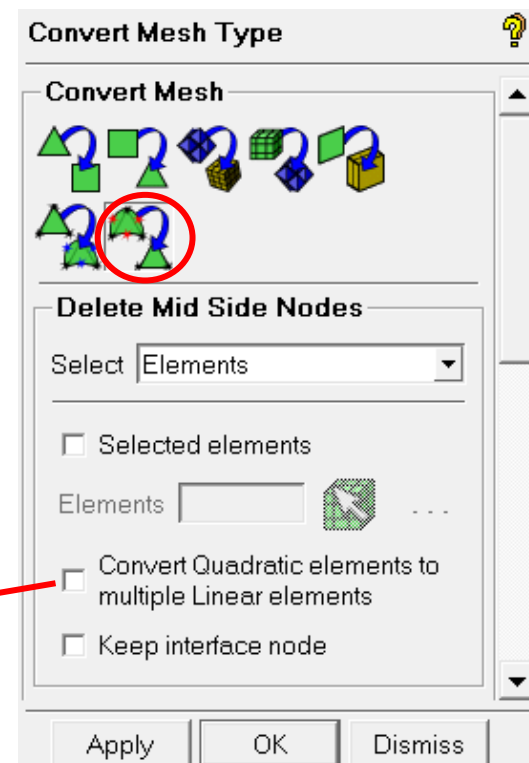
- With the quadratic elements, you would then write the mesh to the structural solver
- But to continue on with this tutorial, you will next convert to a different element type, and this will not work with the quadratic elements, so you must first convert them back to linear
- **Edit Mesh > Convert Mesh Type > Delete Mid Side Nodes**



- **Apply** with the defaults



If **Convert Quadratic elements to multiple Linear elements** is ON, it will split edges at the mid nodes while converting to linear (this is uniform refinement)



Convert to Quad

- First, isolate out just the shells
 - You could do this by deleting all volume elements, but the act of selecting all volume elements can take considerable processor time
 - So it is better to save just the shells to a new file and then load that mesh

File Edit View Info

New Project...
Open Project...
Save Project...
Save Project As...
Close Project...
Change Working Dir...

Geometry

Mesh

Blocking
Attributes
Parameters
Cartesian

Mesh
Subsets
added
Points
Lines
Shells
Volumes

Open Mesh...
Open Mesh Shells Only...
Load from Blocking
Save Mesh...
Save Mesh As...
Save Visible Mesh As...
Save Only Some Mesh As
Close Mesh...

Edit Mesh



- Edit Mesh > Convert Mesh Type > Tri to Quad**
 - Turn on **Mesh** under **Surface Projection**

Convert Mesh Type

Convert Mesh



Tri to Quad

Elements ☐  ...

Surface Projection

☐ Off
☐ Geometry

☒ Mesh

☐ Quadrization

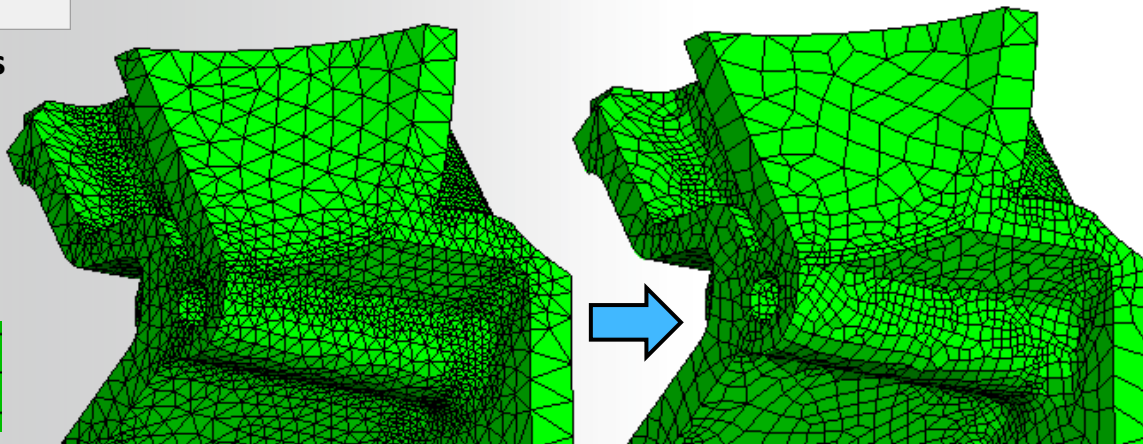
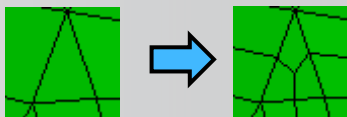
Mesh improvement 1

Max skewness 160

Max warpage 20

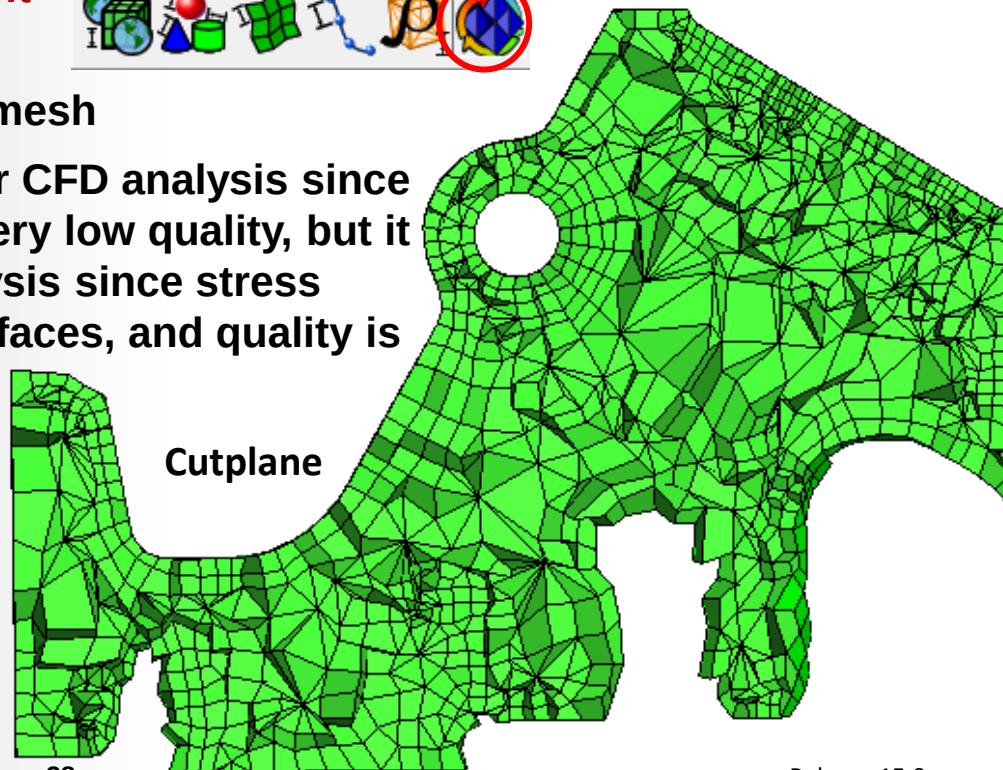
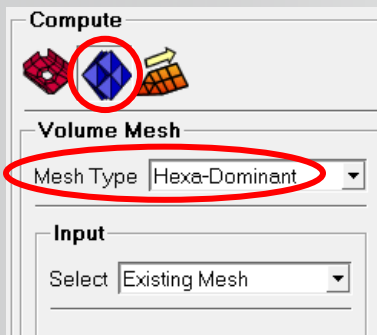
Apply OK Dismiss

Quadrization converts to all quad but makes much more elements by splitting triangle edges in half and propagating the splits



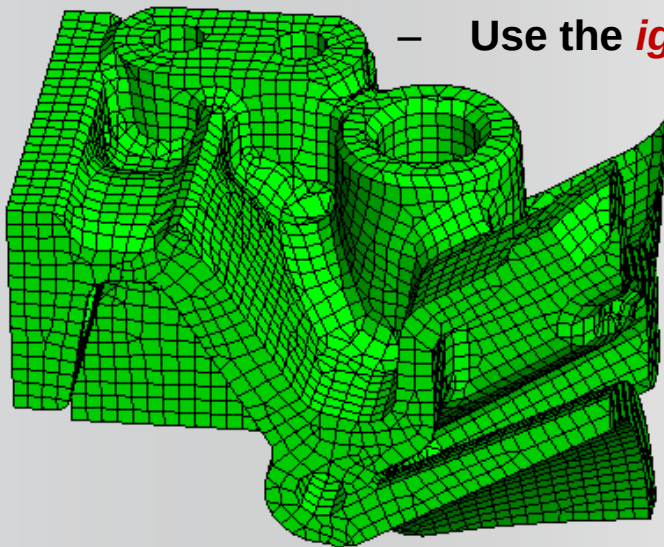
Hexa Dominant

- Hexa dominant is a volume filling mesher which first requires a closed surface mesh of quad dominant shells
- This mesher inflates elements from the surface, and when hexa elements begin to touch, it finishes by filling the center with tetras. Some transition elements (pyramids, prisms) often form at the center also
- **Mesh > Compute Mesh > Volume Mesh**
 - **Mesh Type = Hexa-Dominant**
 - **Input = Existing Mesh**
- View Cutplanes to see internal mesh
- This mesh is not appropriate for CFD analysis since the center elements can have very low quality, but it is fine for many structural analysis since stress concentrations occur at the surfaces, and quality is much better near the surfaces

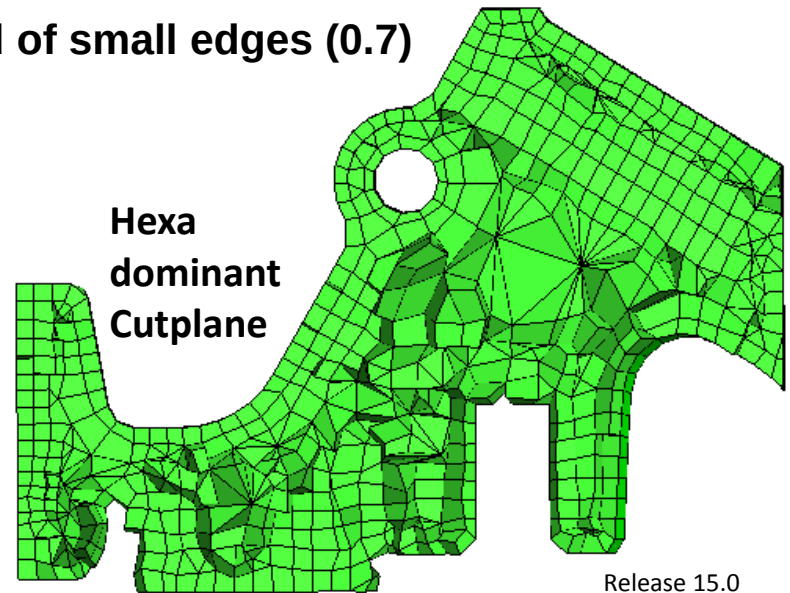


Patch Dependent Surface Mesh

- The previous procedure of meshing with tetra octree, and then saving the surface mesh and converting the tri's to quad's is the same thing as running the **Patch Independent** surface mesher and setting the **Mesh Type** to **Quad Dominant**
- Use the **Patch dependent** surface mesher, then use **Hexa Dominant** filling
 - This results in a better pattern of quads and a better hexa dominant fill
 - The hexa dominant mesher requires a completely closed surface mesh:
 1. Either stitch/match or repair all yellow curves (from build topology)
 2. Or build topology with a higher tolerance (0.13)
 3. Or merge nodes to connect across gaps after surface meshing
 - For patch dependent meshing, you should run build topology with no filtering
 - Use the **ignore size** to get rid of small edges (0.7)



Patch
dependent
surface
mesher



Hexa
dominant
Cutplane