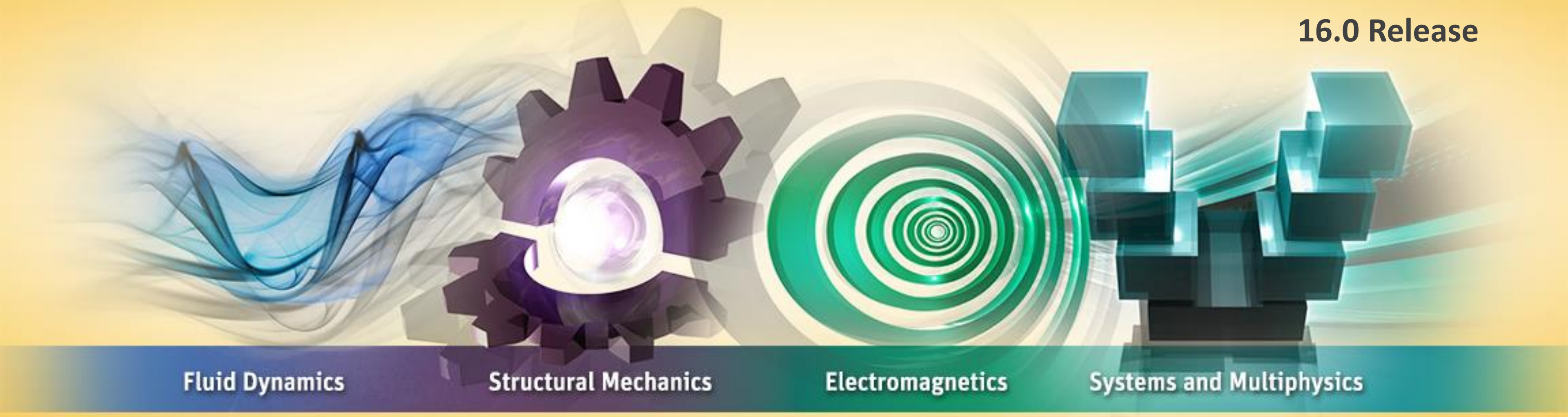
Workshop 11.1
Meshing Evaluation

16.0 Release

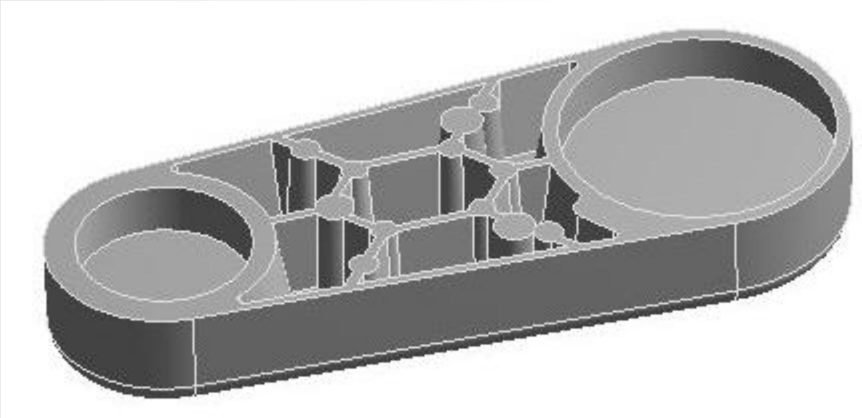


Introduction to ANSYS Mechanical

Goals

In this workshop an arm from a mechanism will be solved using several different meshes for comparison.

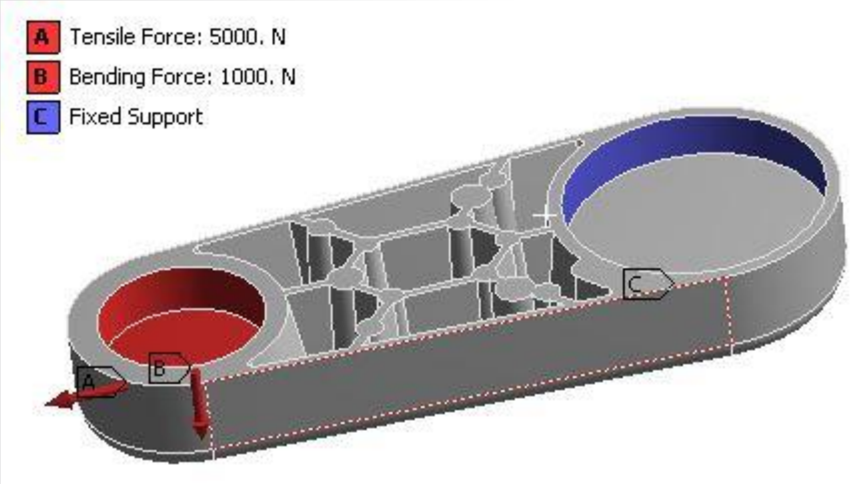
Our goal is to explore how meshing changes can have dramatic effects on the quality of the results obtained.



Assumptions

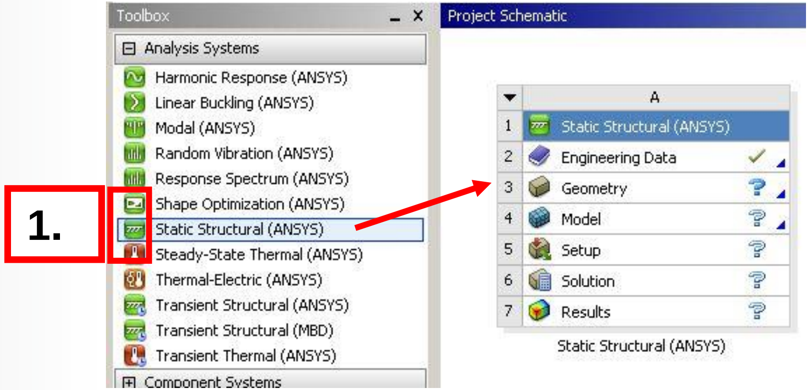
In the loading conditions being simulated the arm is experiencing both tensile and bending loads as shown here.

Our area of interest is the web section that reinforces the interior of the arm.

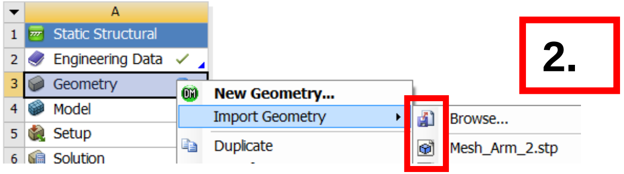


Project Schematic

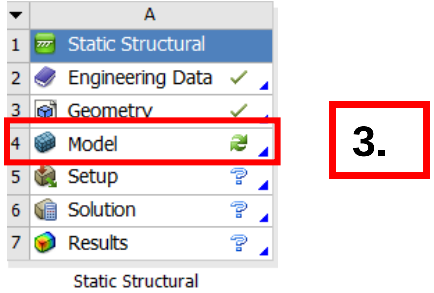
1. From the Toolbox double click "Static Structural" to create a new system.



2. RMB the geometry cell and "Import Geometry" and browse to "Mesh_Arm_2.stp".



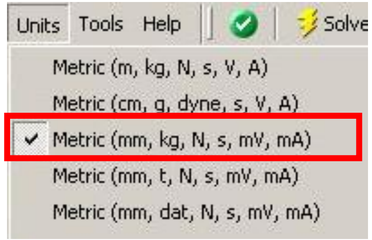
3. Double click the "Model" cell to open the Mechanical application.



Preprocessing

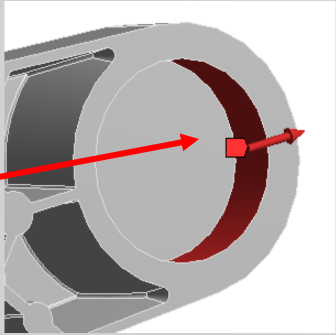
4. **Set the working unit system in Mechanical:**
   – **“Units > Metric (mm, kg, N, s, mV, mA)”.**

4.

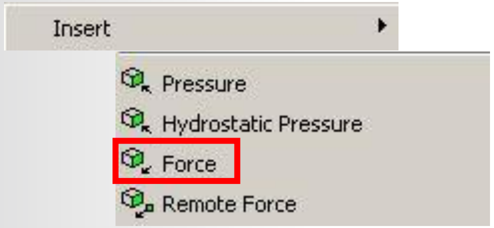


5. **Apply the tensile force on the arm (highlight the static structural branch):**
   a. Highlight the smaller interior cylindrical face.
   b. RMB > Insert > Force
   c. In the detail window choose the component method and enter 5000N in the Y direction.

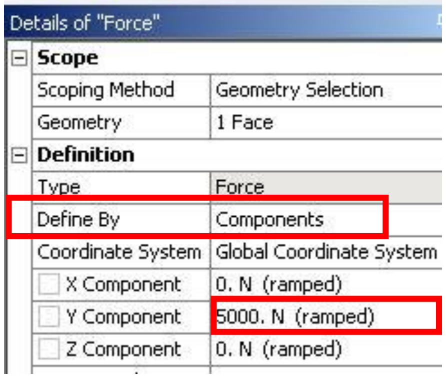
5a.



5b.



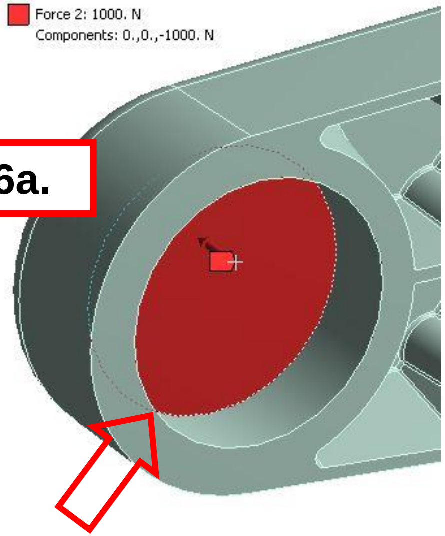
5c.



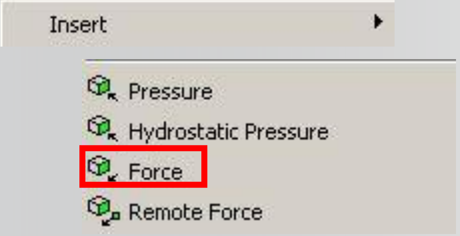
. . . Preprocessing

6. **Apply the bending force to the arm:**
   - a. Highlight the circular face at the base of the smaller end of the arm.
   - b. "RMB > Insert > Force".
   - c. In the detail window choose the component method and enter 1000N in the -Z direction.

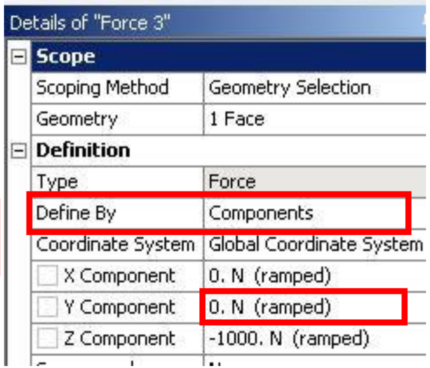
6a.



6b.



6c.



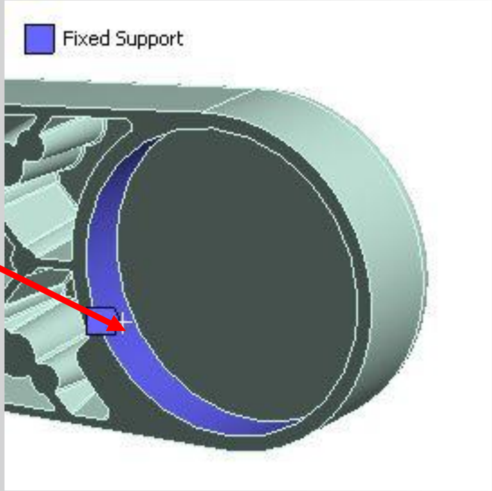
Details of "Force 3"

Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Force
Define By	Components
Coordinate System	Global Coordinate System
X Component	0. N (ramped)
Y Component	0. N (ramped)
Z Component	-1000. N (ramped)

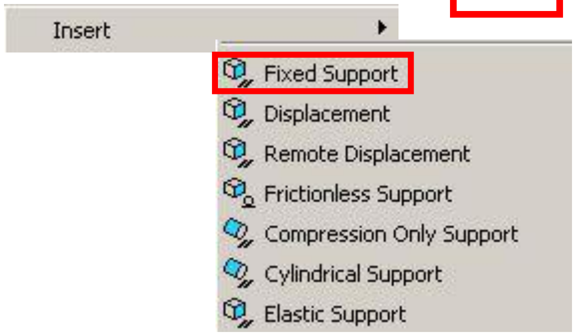
. . . Preprocessing

7. Apply the fixed support on the arm:
   a. Select the larger diameter interior cylindrical face.
   b. RMB > Insert > Fixed Support.

7a.

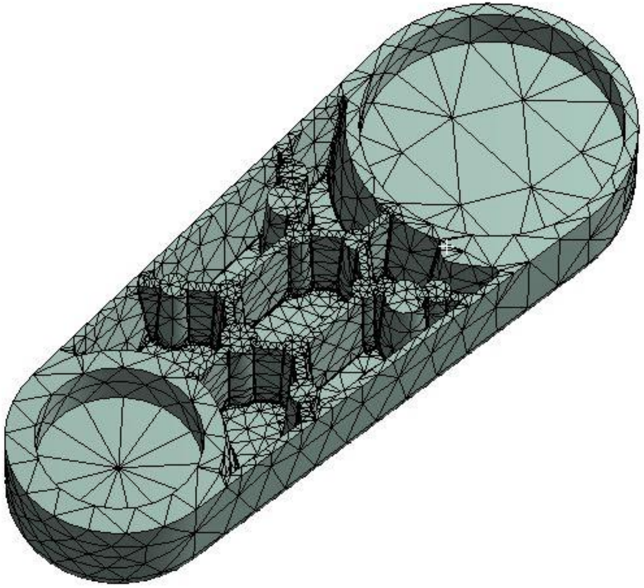


7b.



Meshing

8. **Mesh the arm using all default settings:**
   a. Highlight the mesh branch.
   b. “RMB > Generate Mesh”.

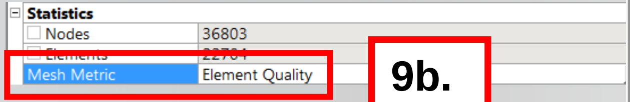


Inspection of the completed mesh shows a very coarse result. In real applications we would likely refine the mesh before solving.

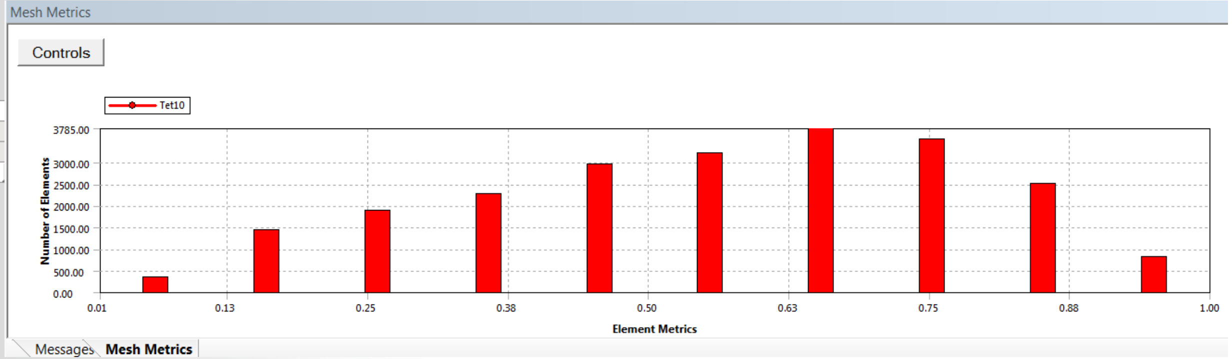
. . . Meshing

9. **Check the element quality:**
   a. **Highlight the Mesh branch.**
   b. **In the Statistics details set "Mesh Metric" to "Element Quality".**

9a.



9b.

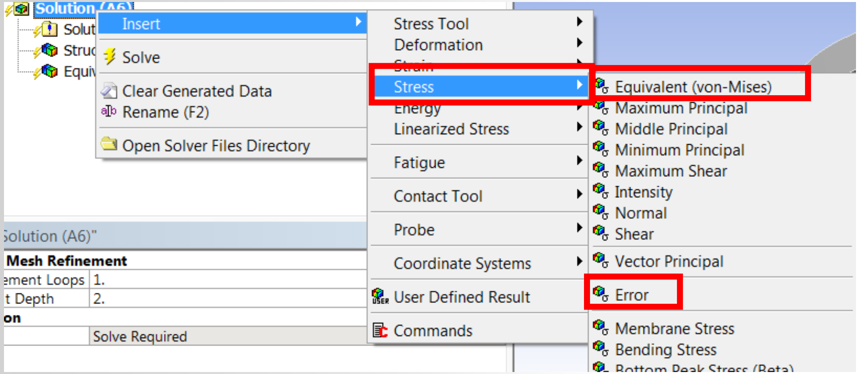


The element quality plot shows that some elements are of a relatively low quality. However, to illustrate some of the practices and tools we'll solve the model as it is.

Solution

10. **Request Results:**
    a. Highlight the "Solution" branch (A6).
    b. RMB > Insert > Stress > Equivalent Von Mises Stress.
    c. RMB > Insert > Stress > Error.

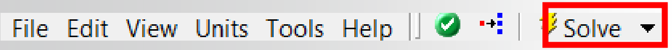
10a.



10b.

10c.

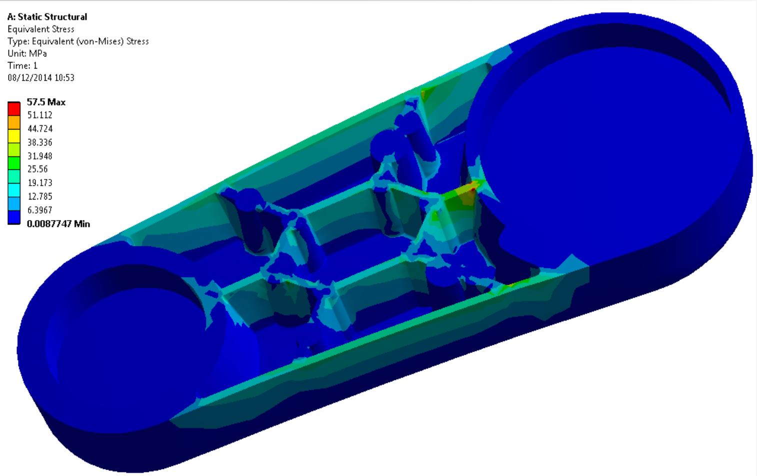
11. **Solve the model:**
    a. Click Solve.

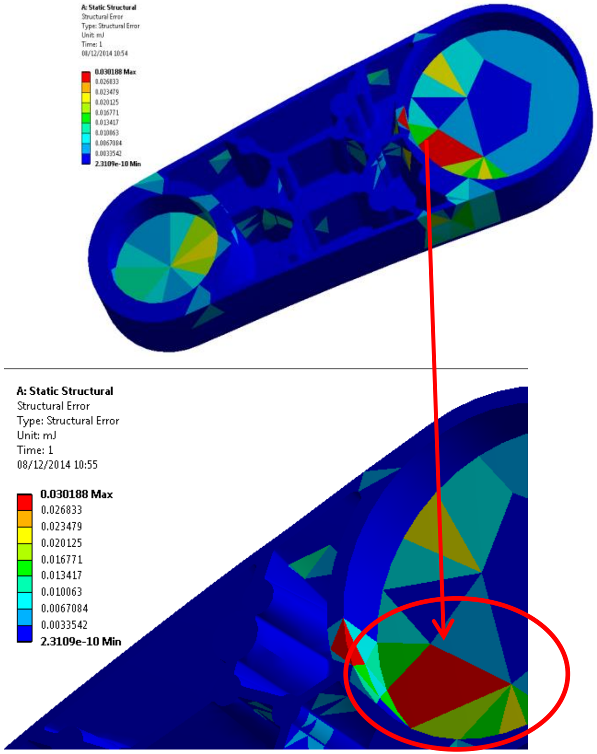


11a.

12. View Initial Results:

The stress result shows the web sections may be an area of concern. In reviewing the error plot however we confirm there is a rapid transition from high to low energy in adjacent elements. This is an indication that mesh refinement is recommended.





Structural Error

Re-Meshing

At this point there are numerous mesh controls we could employ to improve the mesh. We'll focus on the potential problem area indicated in the results using several meshing controls.

13. Change the global mesh settings:

  a. Highlight the mesh branch.

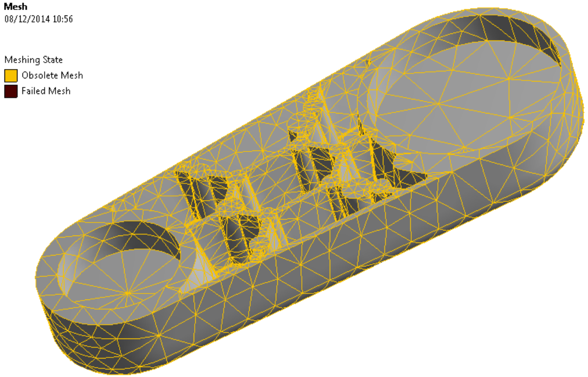
  b. In the details change the "Relevance Center" to "Medium".

Details of "Mesh"

Defaults	
Physics Preference	Mechanical
Relevance	0
Sizing	
Use Advanced Size Function	Off
Relevance Center	Medium
Element Size	Default
Initial Size Seed	Active Assembly
Smoothing	Medium
Transition	Fast
Span Angle Center	Coarse
Minimum Edge Length	1.1115e-002 mm

13b

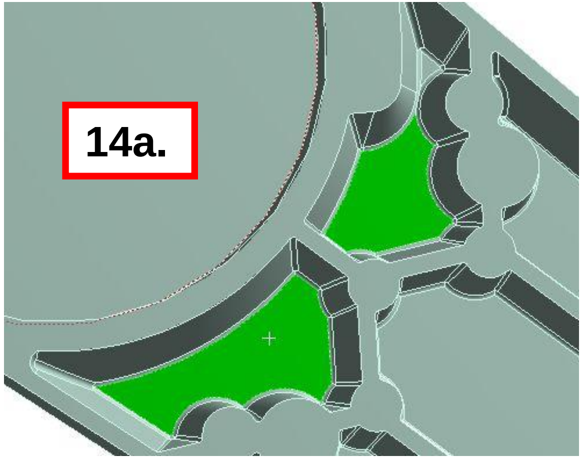
13a.

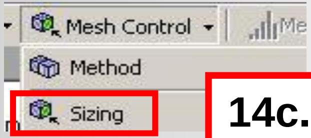


Note: We obtain visual input on any failed, and/or mesh that is not active/obsolete.
This occurs when we update the mesh settings.

. . . Re-Meshing

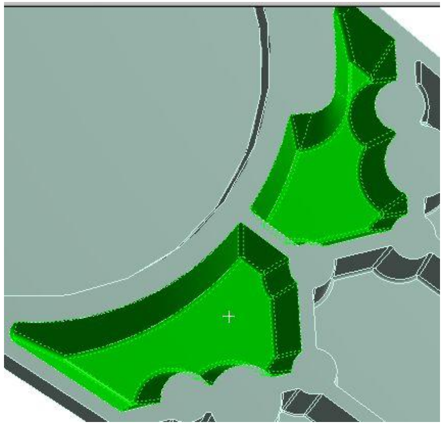
14. Add a mesh size control:
    a. Highlight the 2 faces at the bottom of the cavity (shown at right).
    b. Choose “Extend to Limits”.
    c. RMB > Insert > Sizing.
    d. In the details set the element size to 3mm.





14c.

14b.



Details of "Face Sizing" - Sizing

Scope	
Scoping Method	Geometry Selection
Geometry	72 Faces
Definition	
Suppressed	No
Type	Element Size
Element Size	3. mm
Behavior	Soft

14d.

Note: extend to limits should result in 72 faces being selected.

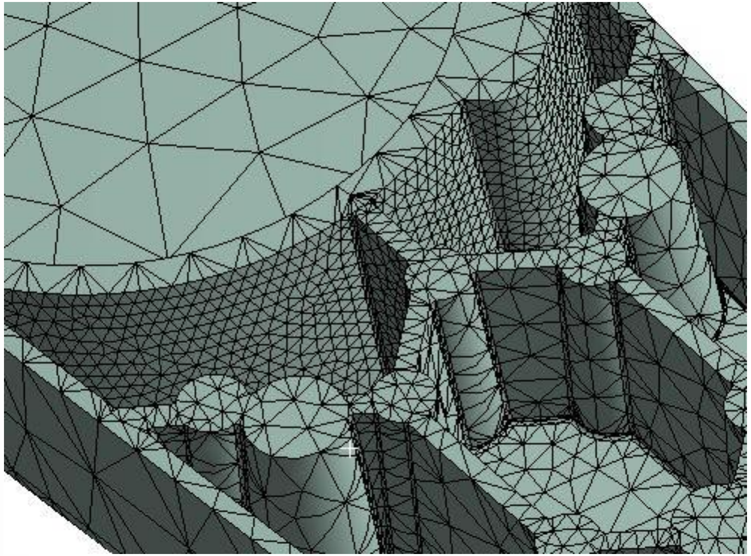
. . . Re-Meshing

15. Remesh the model:
    a. Highlight the “Mesh” branch.
    b. RMB > Generate Mesh.

The new mesh shows we’ve accomplished refinement around the region of interest.

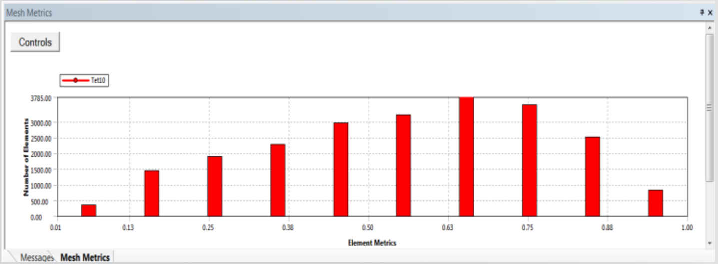
15a.

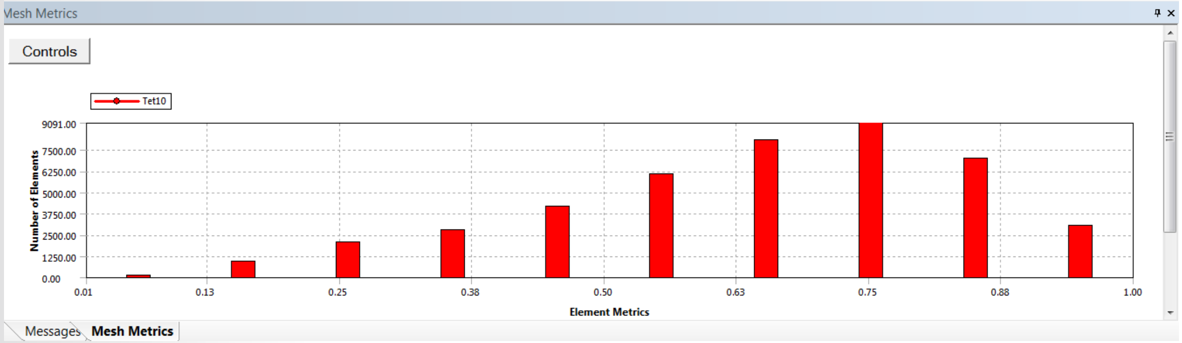
15b.



. . . Re-Meshing

16. **Again reviewing the element quality metric from the mesh statistics detail an improvement can be seen.**

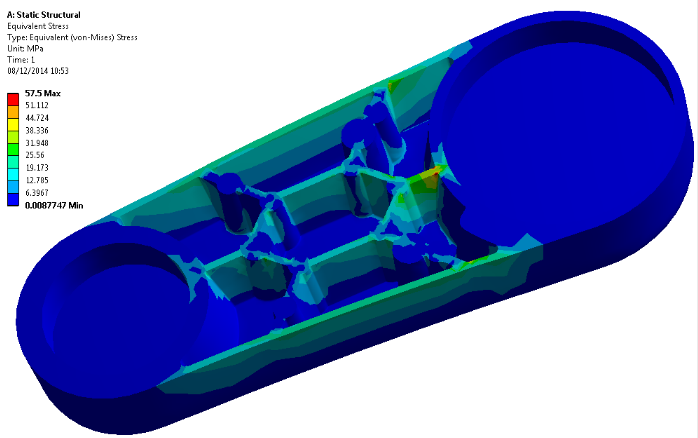




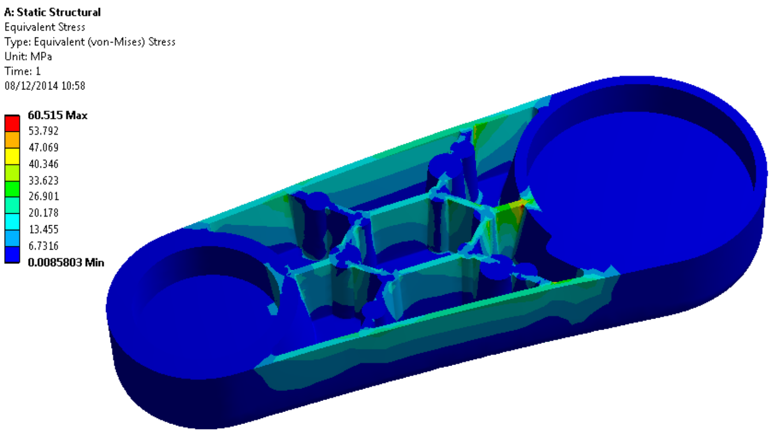
Original Mesh **Refined Mesh**

Results

- Solve the model and review results as before.
- A comparison of stresses from our original mesh shows the maximum value has gone from approximately 57.5 to 60.515 MPa



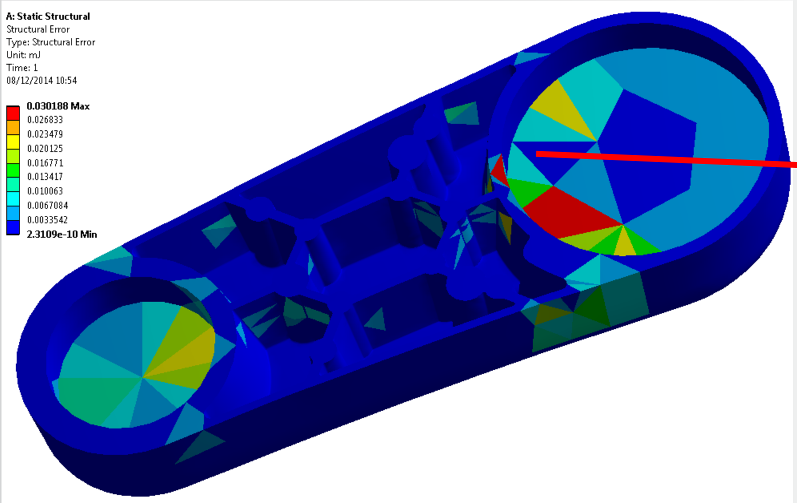
Original Mesh

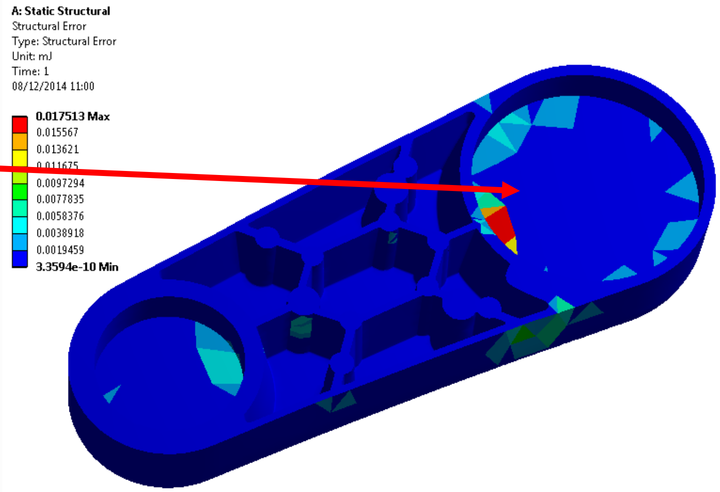


Refined Mesh

. . . Results

Compare error plots for the region of interest.





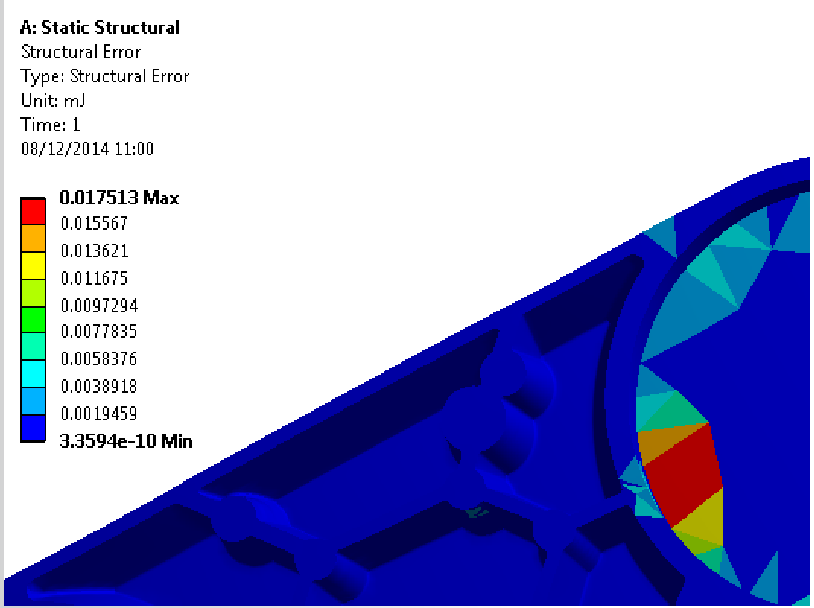
Original Mesh

Refined Mesh

- It’s clear the refinement has reduced the rapid transition in energy values when compared to the original mesh.

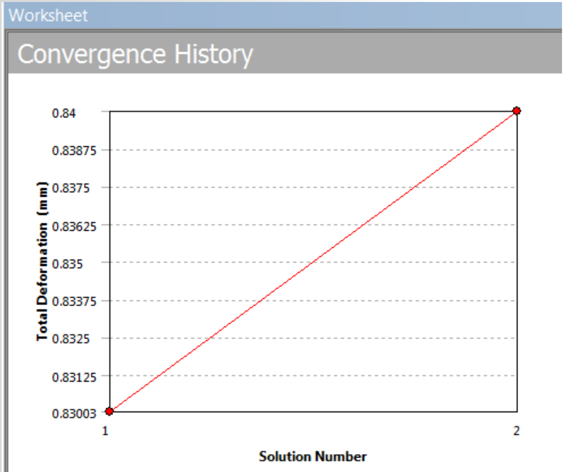
Conclusion

Notice that there are still areas of high energy transition in the model. Our mesh refinement has addressed our stated goal but not the entire model. Each simulation is unique and will require different approaches to insure high quality results.



Go further!

- **Utilising the mesh convergence tool. Obtain convergence of the displacement of the structure to within 10% allowable change.**



	Total Deformation (mm)	Change (%)	Nodes	Elements
1	0.83003		36803	22704
2	0.84	1.1936	103100	68837

