

Spacecraft Assembly - Verification & Validation

Authors: Rajesh Bhaskaran & Michael Mahoney, Cornell University

Problem Specification

1. Pre-Analysis & Start-Up

2. Geometry

3. Mesh

4. Physics Setup

5. Numerical Solution

6. Numerical Results

7. Verification & Validation

Exercises

Comments

Verification & Validation

We do not have experimental data to validate our results. So we'll be carrying out only verification here.

Comparison with Hand Calculations

We'll check the following using hand calculations:

- Reaction in axial (z) direction where mid nozzle is attached to top nozzle (not modeled)
- Thermal strain
- Hoop stress
- Bolt preload

Reaction in Axial Direction

The reaction reported by ANSYS is within 10% of the hand calculation as per the images below.

Hand Calculations: Reaction in Axial (z) Direction

- Average gas pressure:

$$\rightarrow \frac{12.17 + 47.72}{2} \approx 30 \text{ psi}$$

- Top radius = 41.75 inches

Bottom radius = 69.50 inches

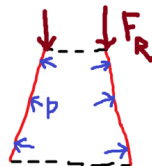
- Projected area in z direction =

$$\pi(69.50^2 - 41.75^2) = 9699 \text{ in}^2$$

- Net pressure force in z direction = $30 * 9699$

- Net reaction force in -z direction on 1/400th

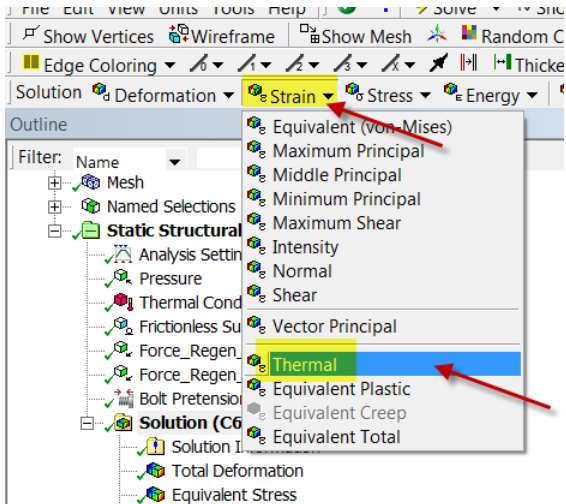
$$\text{model} = \frac{30 * 9699}{400} \approx 720 \text{ lbf}$$



Details of "Force Reaction"	
Options	
Result Selection	All
☐ Display Time	End Time
Results	
☐ X Axis	0.66212 lbf
☐ Y Axis	0.10593 lbf
☐ Z Axis	-683.68 lbf
☐ Total	683.68 lbf
Maximum Value Over Time	
☐ X Axis	0.66212 lbf
☐ Y Axis	0.10593 lbf
☐ Z Axis	-3.8343e-006 lbf
☐ Total	683.68 lbf
Minimum Value Over Time	

Thermal Strain

Select *Solution > Strain > Thermal*

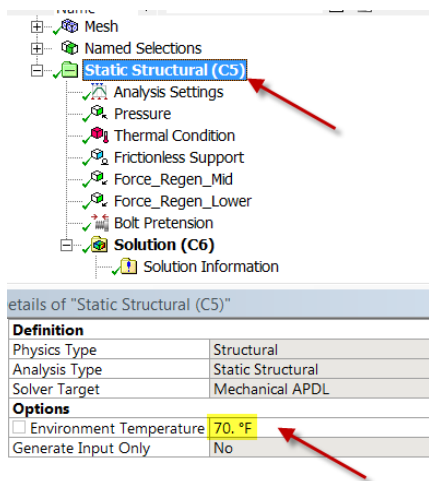


Check that the thermal strain reported by ANSYS for all parts compares well with the expected value from $\alpha \cdot (dT)$.

dT is calculated as $T_2 - T_1$

T_2 is the temperature specified in the *Thermal Condition* under *Static Structural*. Note that α is calculated at this temperature.

T_1 is the *Environment Temperature*. Make sure it is set to 70F as per below.

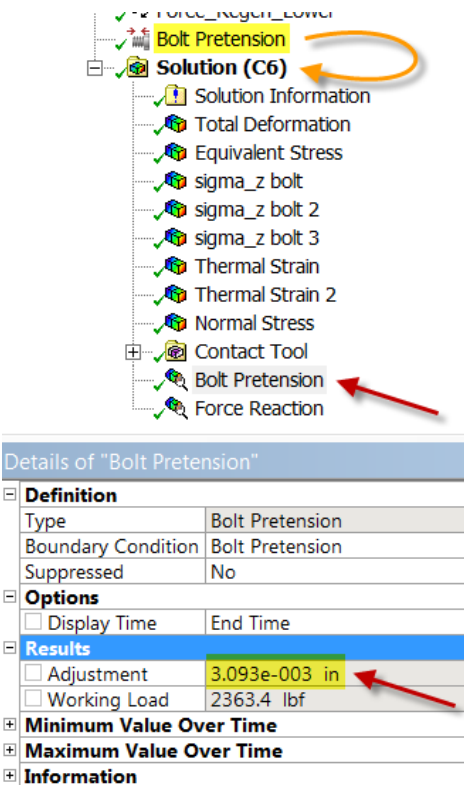


Hoop Stress

The hoop stress can be estimated using $p r/t$. Check this against the corresponding ANSYS result which can be determined by looking at the stress in a direction normal to the symmetry planes. You can spot check at a couple of locations in the model.

Bolt Preload

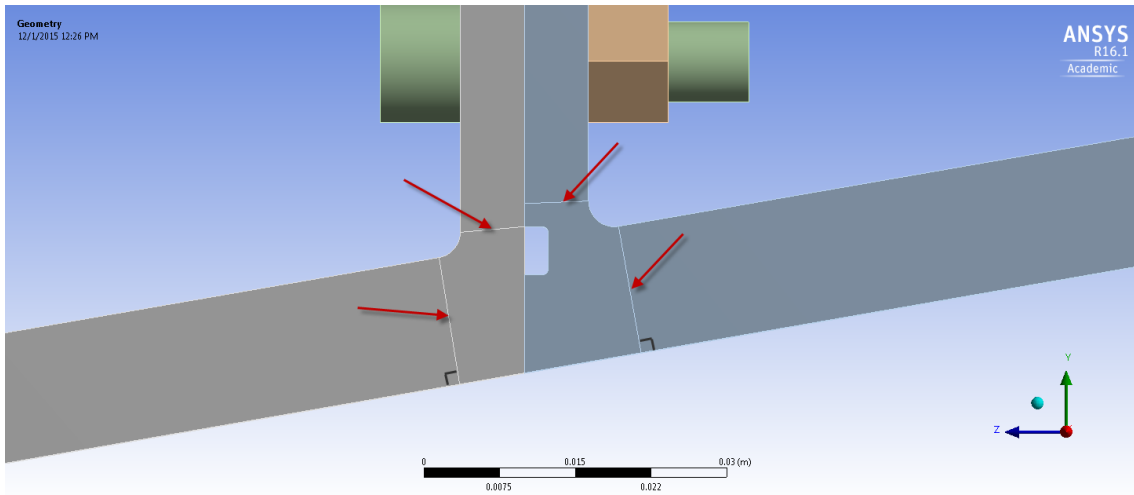
Dragging the *Bolt Pretension* object under *Static Structural* to *Solution* and evaluating results gives the amount by which the bolt has to be shrunk in order to generate the specified preload. See below. The value reported by ANSYS (3.093e-3 in) can be checked using 1D elasticity approximation: $dL = (F L)/(E A)$



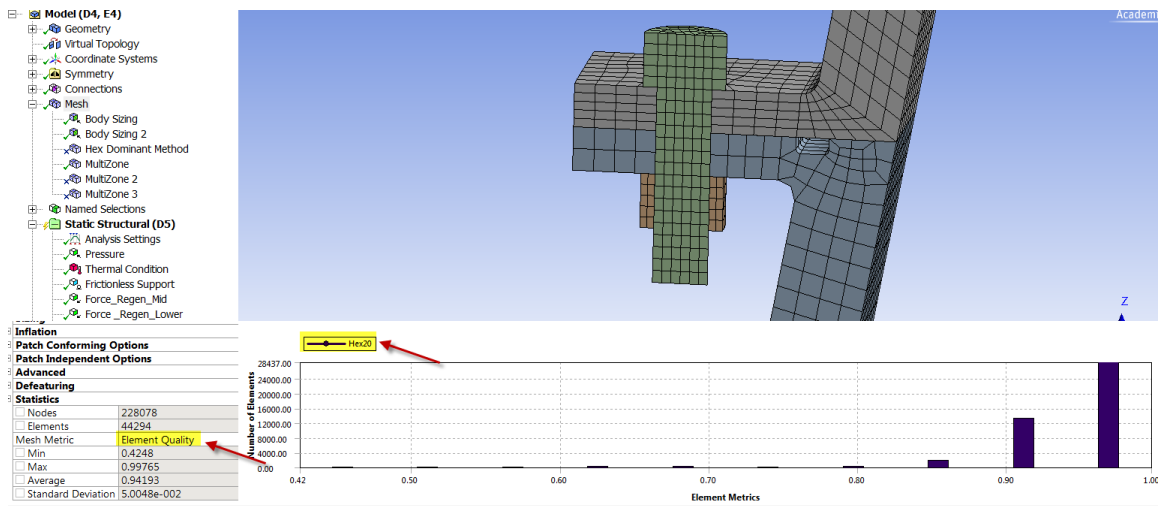
Hex Mesh

The disadvantage of the Hex Dominant meshing technique is that it generates poor elements in transition regions between hex and tet elements. A pure hex mesh can be generated by splitting up the model and using the Multizone meshing technique as per the video below. Heads-up: The solution on the new mesh might take over 5 minutes.

The video considers hex meshing for one part only. It turns out you need to divide the parts as per the image below to get a pure hex mesh for *all* parts. Note that the mesh quality improves if you make two of the splits perpendicular to certain faces as indicated in the image.



Check that you have a pure hex mesh by turning on element quality. The *Hex20* label refers to hex elements with 20 nodes per element including midside nodes.



[Go to Exercises](#)

[Go to all ANSYS Learning Modules](#)