Static Analysis of Rigid Steel Joint For Bending Moment

www.calculixforwin.com

This document is licensed under the Creative Commons Attribution-No Derivative Works 3.0 License. To view a copy of this license, visit http://creativecommons.org/licenses/by-nd/3.0/.

Ref. Movies: | 1 |
Input data: STEP file with geometry (in inches); material: steel, bending moment = 30000 lb*in, fixed end.

Static analysis in ANSYS

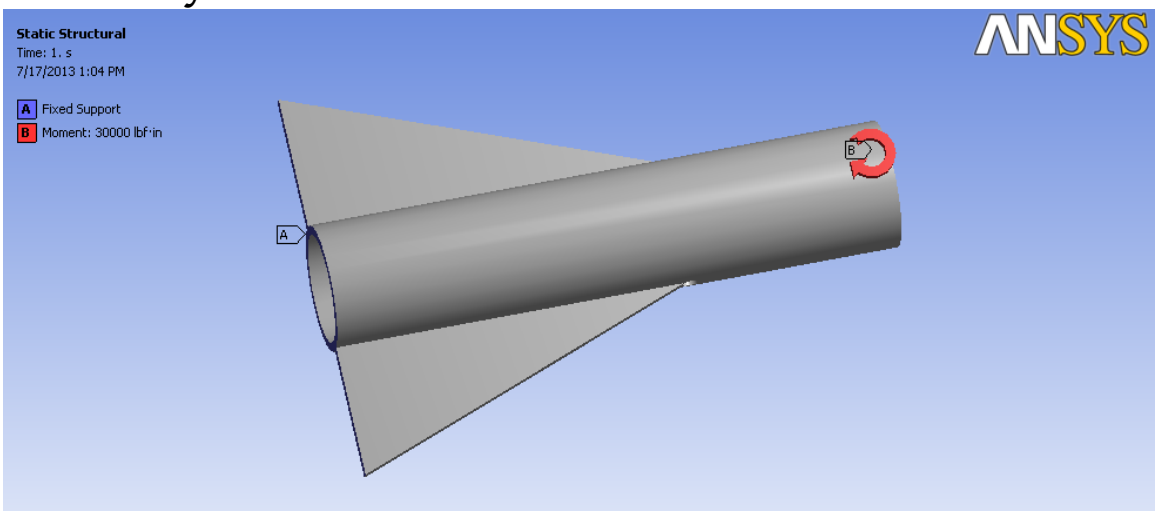

Fig.1 Load&BC Applied

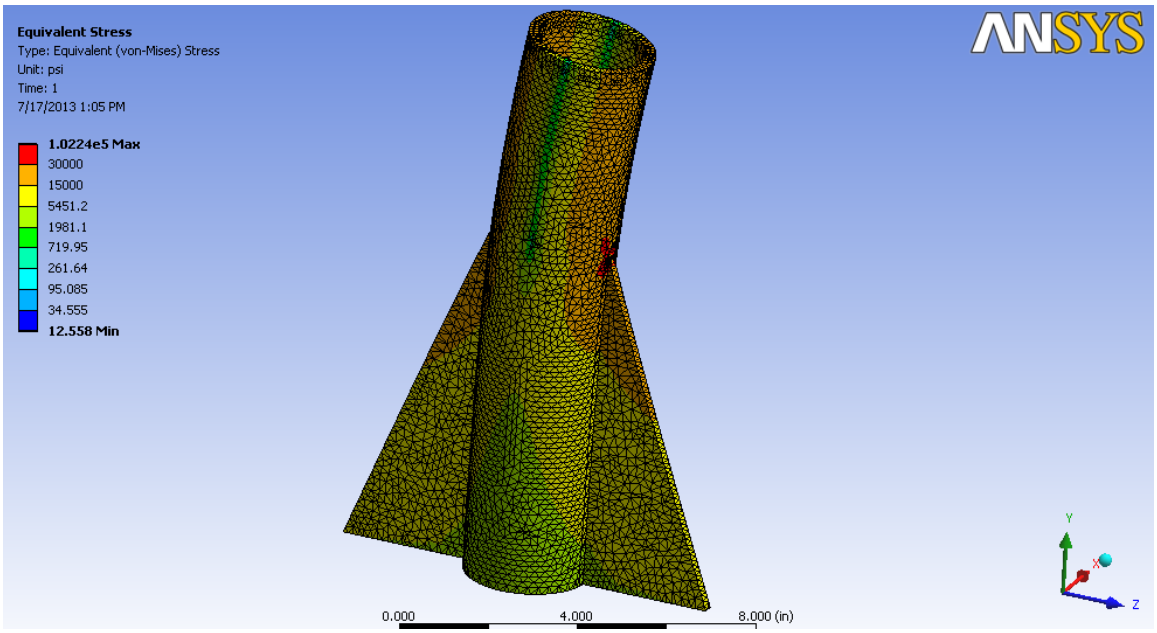

Fig. 2 Von-Mises Stress, psi, (the max. value at singularity point is not determined)

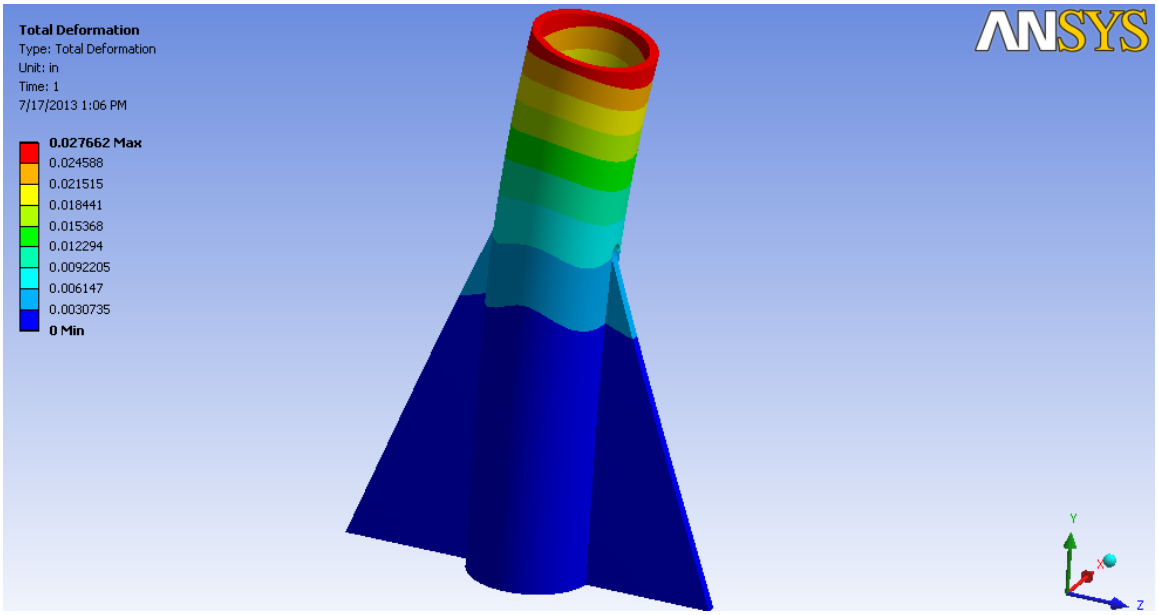

Fig. 3 Displacements, in

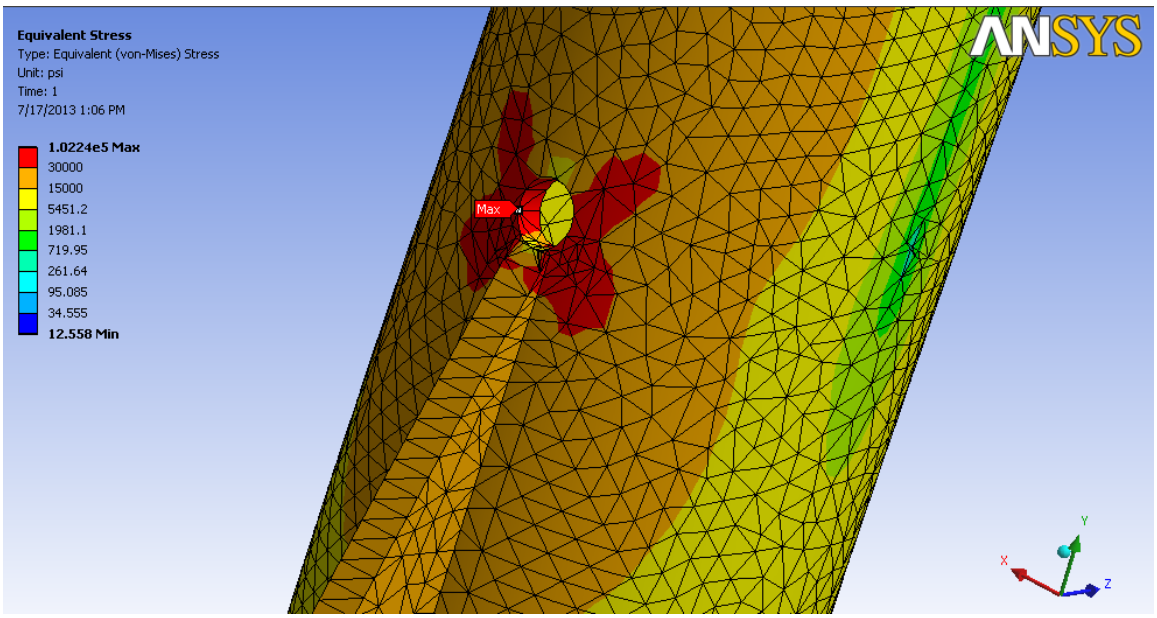

Fig. 4 Von-Mises Stress Distribution (more than 30 ksi in red)

Main Steps:

1. Run Netgen exe (c4w\programs\Netgen\bin\) and load geometry in step file
2. Change meshing options (fig. 5) to make more elements at curved parts (rough quadratic elements may cause negative Jacobian. Also you can export the mesh to GMSH format and check Jacobian with plug-in at GMSH program). Mesh model and set up "second order" after meshing (Refinement>Second Order). Save mesh as .vol file.
3. Run c4w.exe and type the name of problem. Select .vol mesh file and press "Continue>".
4. Scale the mesh from "mm" to "in" and check distance with "qdis" card.

The material was applied automatically after running the model.

4. Update list of groups and find (double clicking) the nodal groups you need to apply the boundary conditions (moment and rigid support).
5. When BC is applied, save "allinone.inp" file with mesh and groups. Run the solution. For large models you may need 64 bit version (replace ccx.exe. with bconverged.com CalculiX freeware). Don't replace CGX executable!. Run Post-processor when job is done.

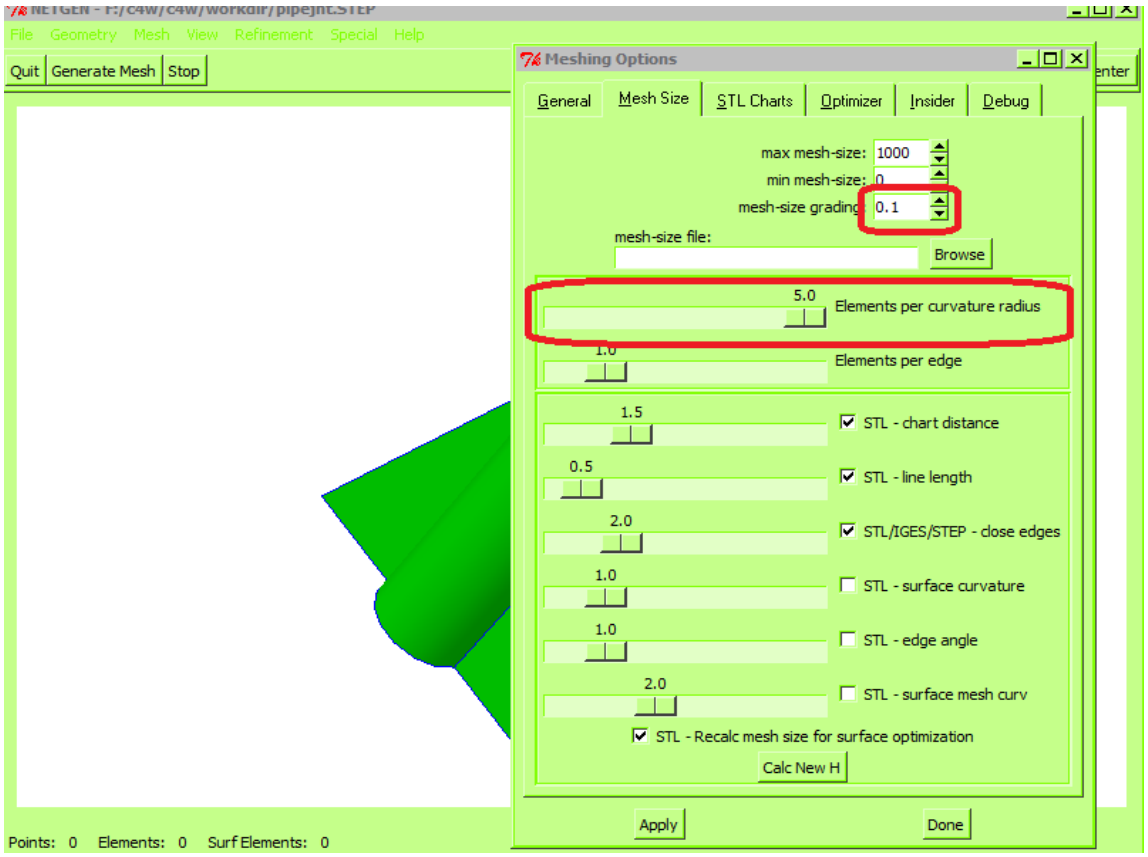

Fig. 5 Netgen Meshing Options

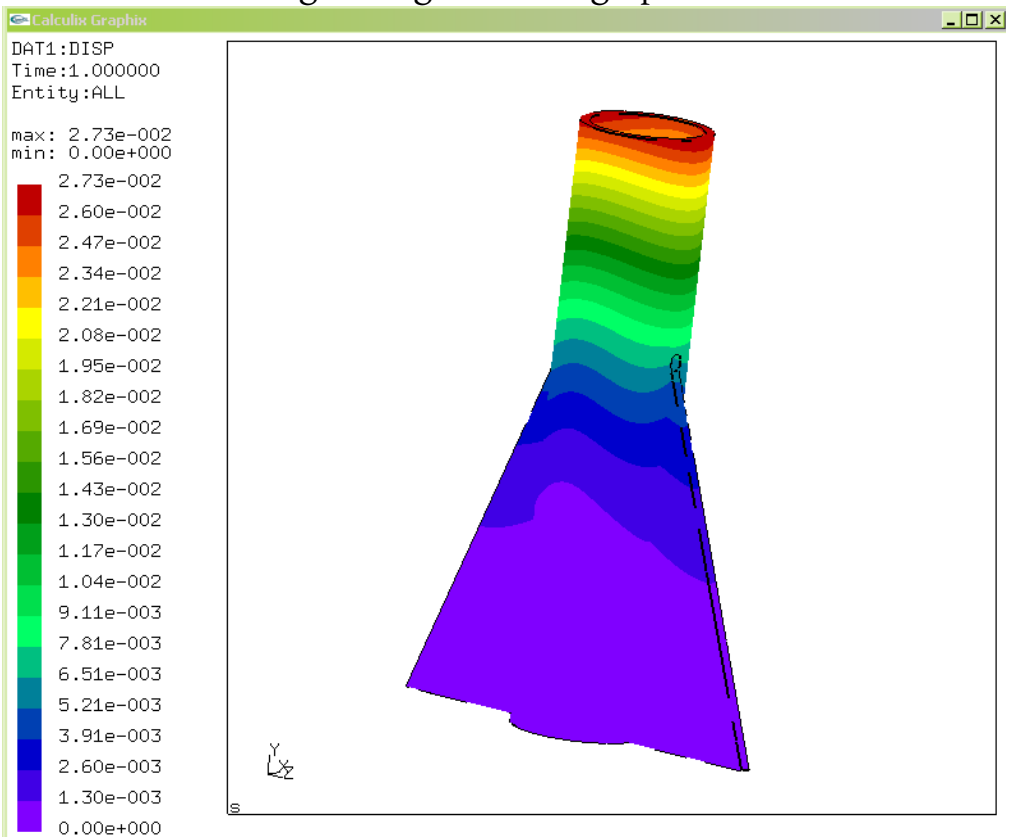

Fig. 6 Max. Displacements, in (0.0273")

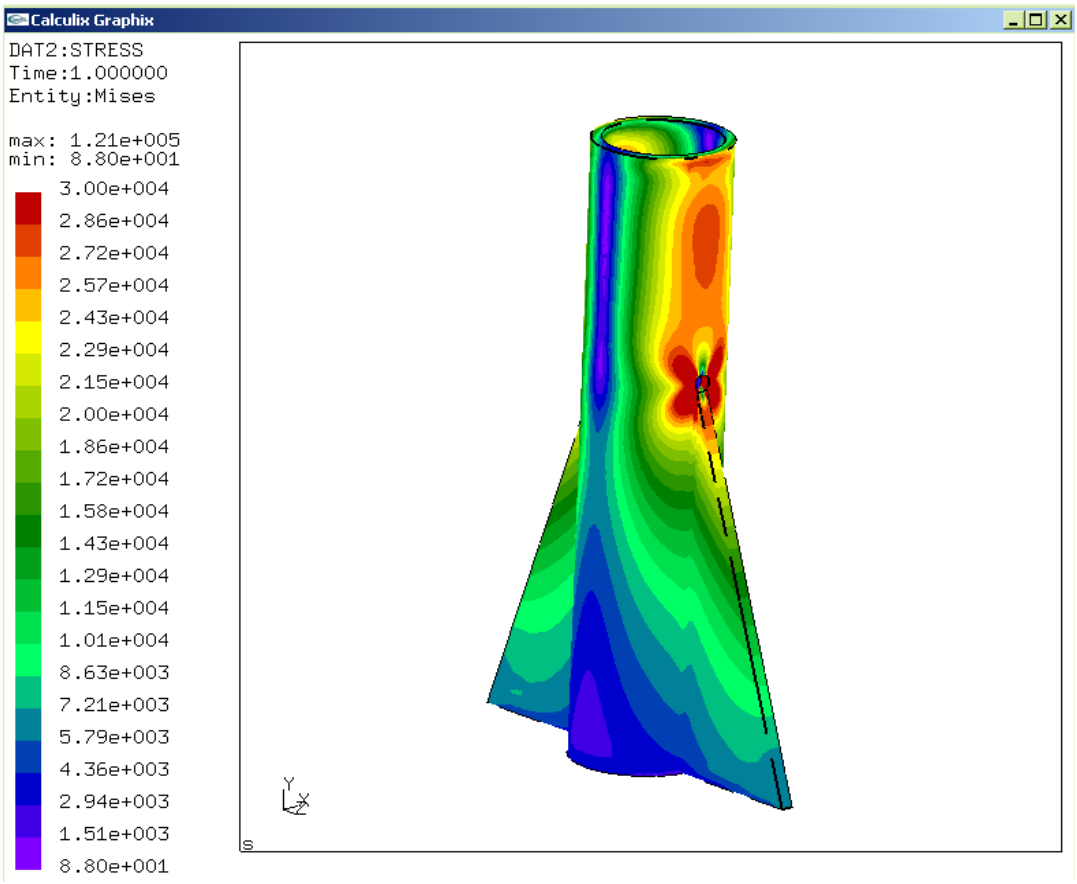

Fig. 7 Von-Mises Stress Distribution (max at 30 ksi)

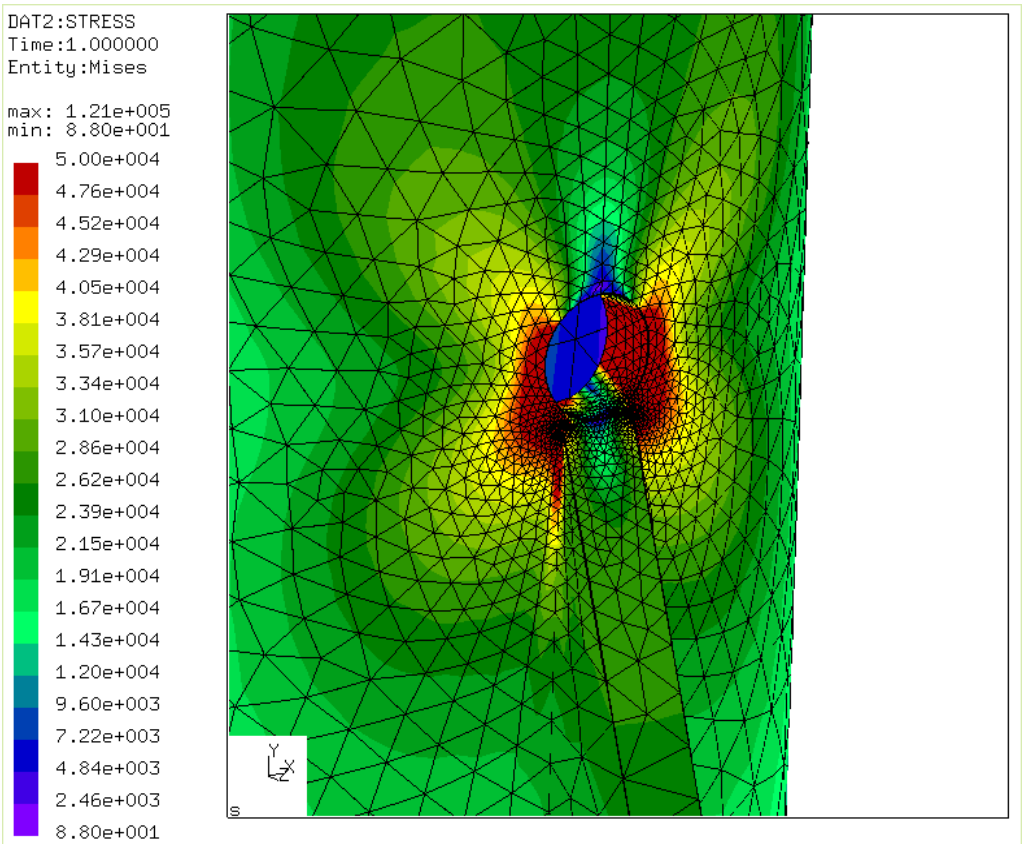

Fig. 8 Von-Mises Stress Distribution (max, at 50 ksi)