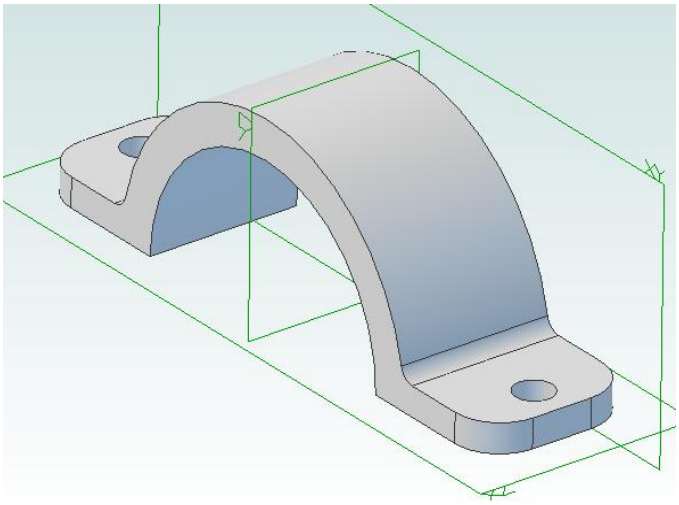
Using FEMdesigner AD

Elastic/Plastic Stress Analysis Tutorial

This tutorial assumes the user has a working knowledge of FEMdesigner AD, as taught by the "Getting Started with FEMdesigner AD Cantilever Beam Tutorial." Please see the support page of our web site to download these and other tutorials (www.femdesigner.com/support.html).

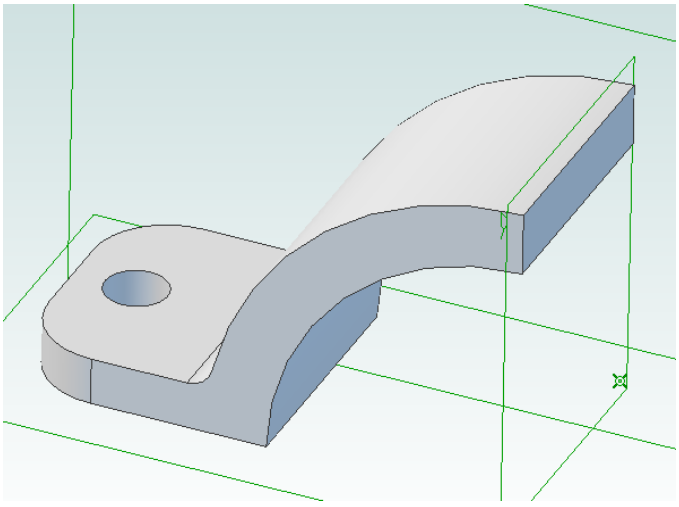
In this elastic/plastic analysis, we will show how to analyze a pipe clamp that is loaded past its yield stress to find the regions of the part that remain in the elastic stress regime, and more importantly, those regions that will experience plastic deformation. We will use part symmetry to simplify the analysis, and will also visualize the internal stresses in the part by dynamically slicing the results mesh.



Part Symmetry

The first thing we notice from an analysis point of view is that this is a perfect example of part symmetry that can be used to our advantage to shorten analysis times.

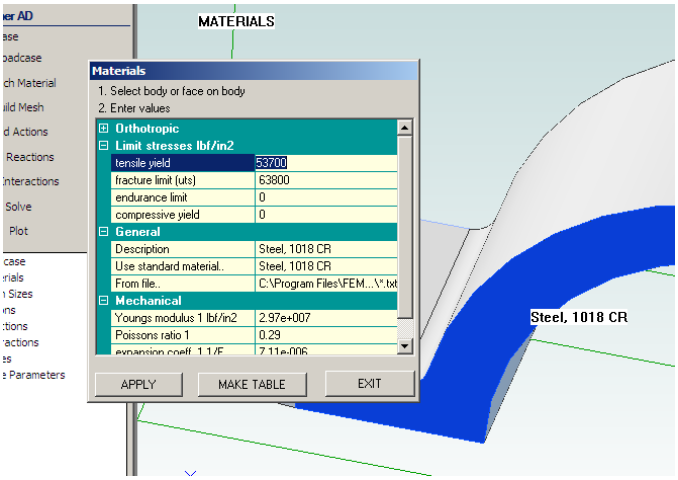
We only need half of this part, properly constrained, to perform an accurate analysis, since both halves will behave in the same way.



Open Part File

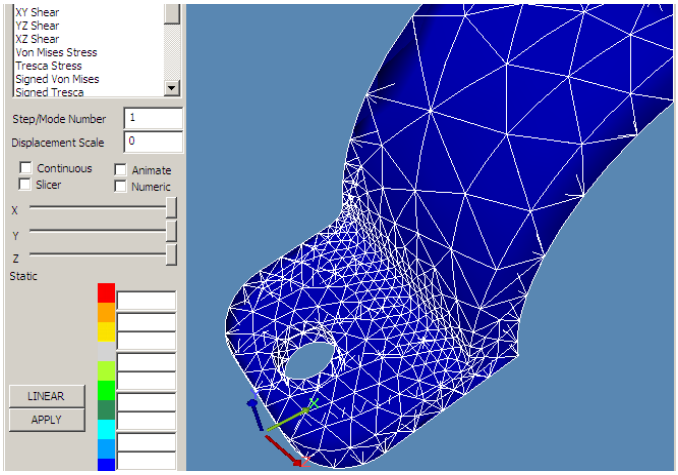
Open the file named halfclamp.AD_PRT that was included in the ZIP file with this tutorial document.

Using half the part, our mesh is half the size, which will translate into *at least* doubling the speed of the solution. Also, loading and restraining will take less time, allowing us to perform more design/analysis iterations and come up with a better quality design using less material.



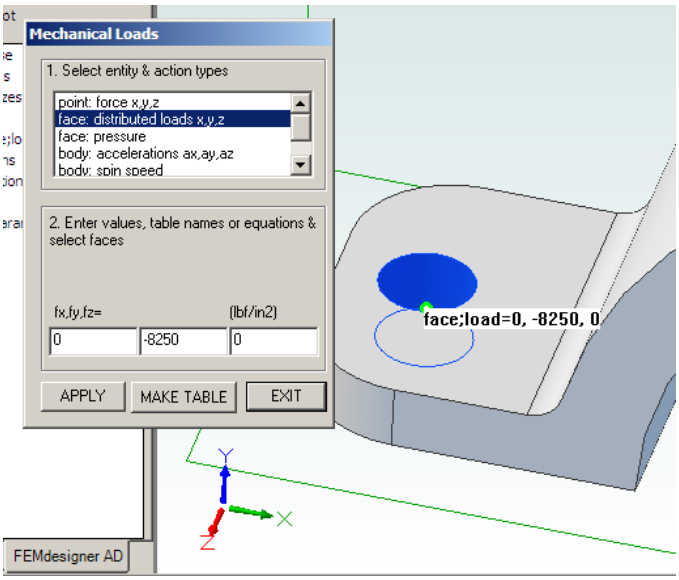
Assign Material

- Launch FEMdesigner AD from the Alibre dropdown menu
- Assign 1018 cold-rolled steel to the part
- Expand the “Limit stresses” section and make sure the tensile yield has a reasonable value, in this case we’ll use 53,700 psi



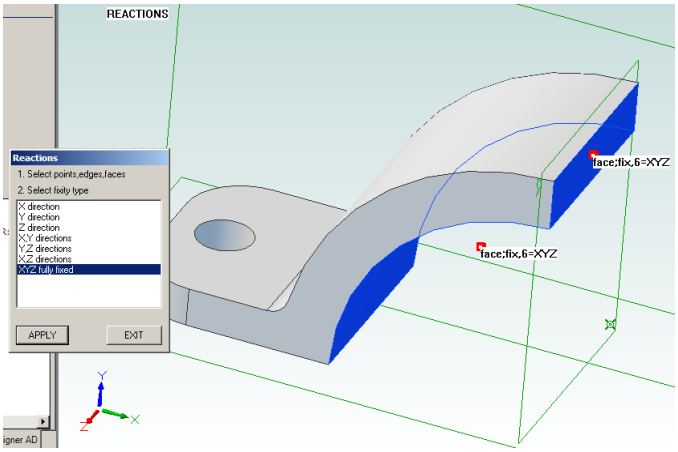
Mesh Part

- Select “Mesh” from the explorer menu and accept the defaults
- Plot mesh to confirm appropriate mesh density



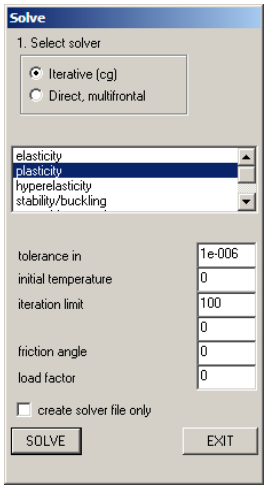
Assign Load

- Using “Actions,” load the inside face of the hole with a distributed x,y,z load
- Load values (psi):
    - X = 0
    - Y = -8250
    - Z = 0
- The negative value for the Y load is important



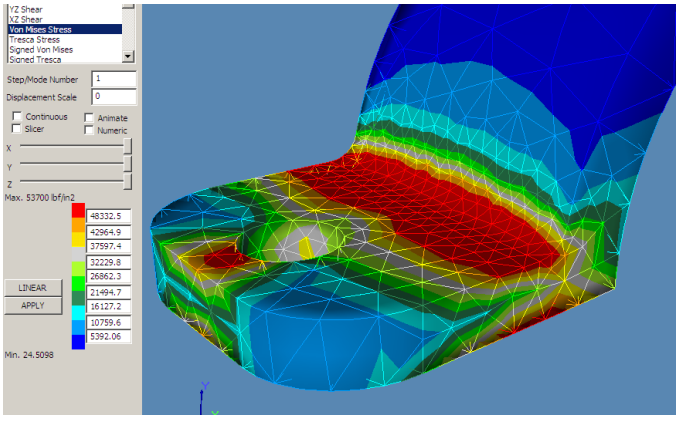
Assign Restraints

- Using “Reactions,” restrain the inside face of the clamp, and also the ‘cut’ face on the plane of symmetry, as shown
- Restraint should be “XYZ fully fixed”



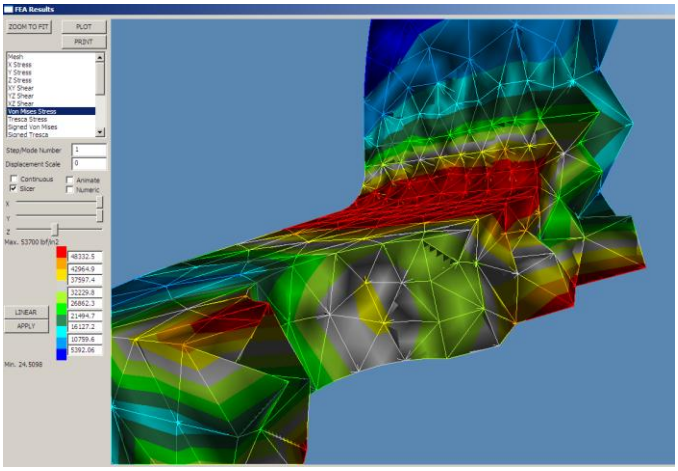
Solve for Plasticity

- Select “Solve” in the FEA explorer
- In the Solve dialog, select “Iterative (cg)” and “plasticity” as shown
- Change error tolerance if desired, and enter 100 for “iteration limit”
- Leave other fields at “0”



View Results

- Select “Plot”
- Select “Von Mises Stress” then click the PLOT button
- Notice that the maximum stress shown in the color-coded stress scale is the yield stress
- All red-colored areas are experiencing plastic deformation



View Internal Stress

We can easily visualize the internal stresses inside parts also. In this case, we may be concerned with how "deep" the plastic stress goes:

- Zoom/spin/pan to approximately the view shown at left
- Click the "Slicer" checkbox
- Move the "Z" slider to the middle of the scale to see the interior stresses
- Can slice the part this way in all three orthogonal directions

Review

In this elastic/plastic analysis tutorial, we modeled the elastic/perfectly plastic behavior of a part stressed past its yield stress. We used part symmetry to simplify the analysis, and we were able to visualize the regions of plastic deformation both on the surface and inside the part's volume.

Thanks for choosing FEMdesigner AD!