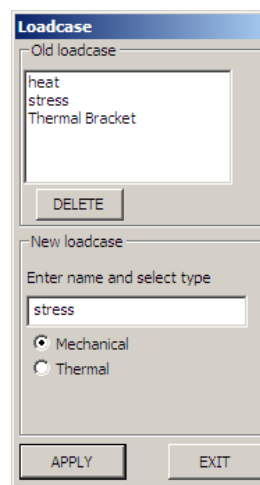# Using FEMdesigner AD
Heat Stress Tutorial

This tutorial assumes the user has a working knowledge of FEMdesigner AD, as taught by the "Getting Started with FEMdesigner AD Cantilever Beam Tutorial." It also assumes that the user has worked through the Steady State Heat Transfer Tutorial. Please see the support page of our web site to download these and other tutorials (www.femdesigner.com/support.html).
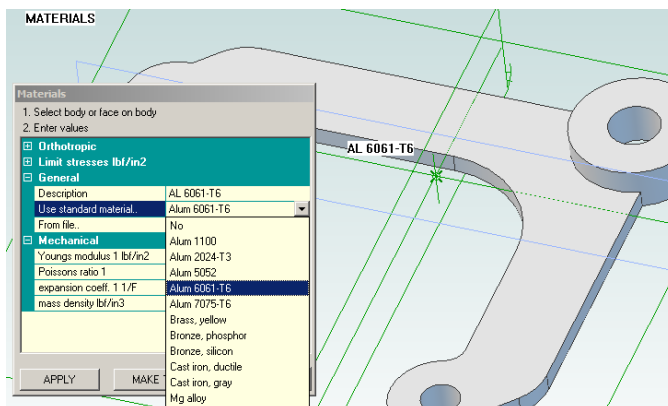
In this heat stress tutorial, we will use the temperature distribution results from the Steady State Heat Transfer Tutorial to calculate stresses induced by thermal expansion of a bracket with unequal length arms by simulating the arms being mounted on fixed shafts.

If you haven't already done so, work through the Steady State Heat Transfer Tutorial, remembering the name of the thermal loadcase you used, and the material you used. For this tutorial we named the loadcase "heat."
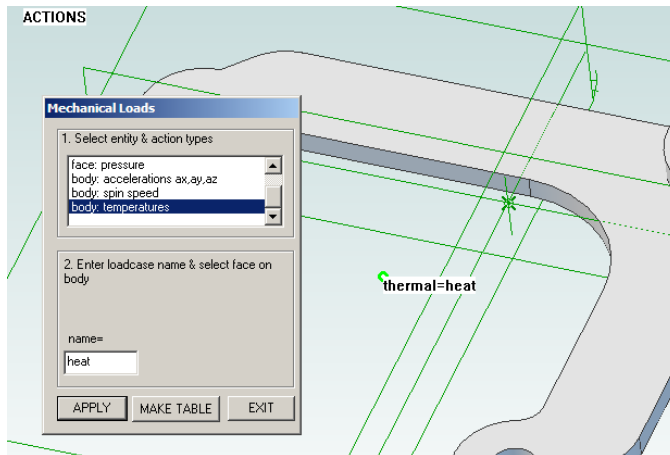

Create New Mechanical Loadcase

- Open Thermal Bracket.AD_PRT and launch FEMdesigner AD from the drop-down menu
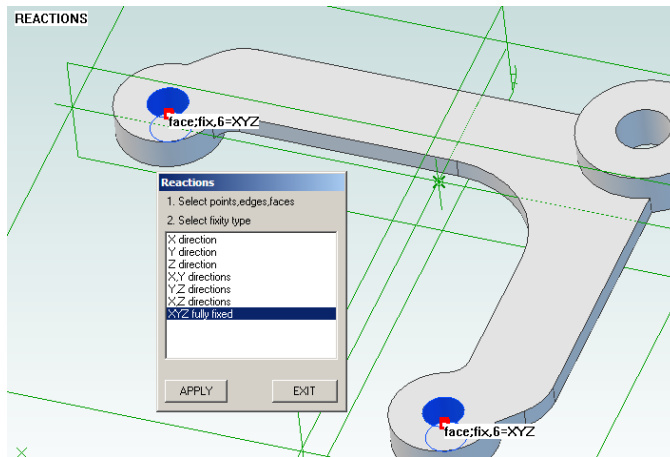- Select "Loadcase," then select "Mechanical," and provide a name and select "APPLY"

Apply Material

- Remember the material used in the Steady State Heat Transfer Tutorial and apply the same material
- While not critical for the purposes of this tutorial, it will be important for the accuracy of your analysis results to use the same material for both the thermal and mechanical portions of a heat stress analysis
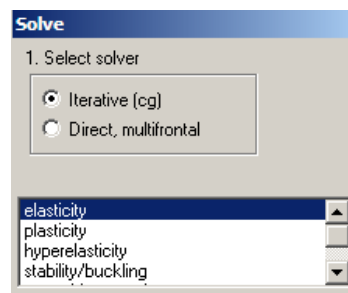
Add Heat Load

- Select "Actions" from the Explorer menu
- Select "body: temperatures" as the load type
- Enter the name of the thermal loadcase from the Steady State Heat Transfer Tutorial, in this case, it is "heat."
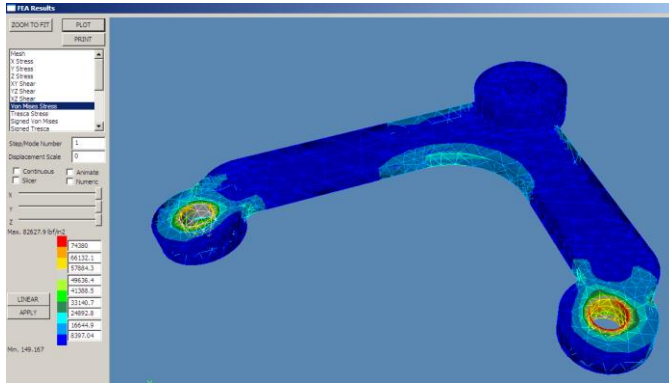

Fix Mounting Bores

To simulate the bracket being mounted on two fixed shafts:

- Select "Reactions" from the Explorer menu
- Select "XYZ fully fixed"
- 'Shift-select' both interior bores at the ends of the arms of the bracket, as shown
- Click "APPLY"

Solve for Heat Stress

- Select "Solve" from the Explorer menu
- Select "Iterative (cg)" and "elasticity"
- Click "SOLVE"

View Results

- Select "Plot" from the Explorer menu
- In the results display window, select "Von Mises Stress"
- Click "PLOT"
- You can also visualize the shape of the deformation induced by the thermal expansion by entering an appropriate Displacement Scale

As a further example that you may want to experiment with, you could add mechanical loads in addition to the thermal load to simulate other conditions the part might experience. If any of these loads create stresses higher than the yield stress, a further plastic analysis would be possible as well.

Review

In this Heat Stress Tutorial, we used the temperature distribution results from a previous steady state thermal analysis as a load to create thermal expansion in our part, then created a mechanical elastic analysis to calculate stresses induced by thermal expansion if the part were mechanically constrained. In order to create this two-level analysis, you have to remember the thermal loadcase used for the initial thermal analysis, as well as the material used for the part to insure accuracy.

Thanks for using FEMdesigner!