**FEMdesigner™**
Thoughtful analysis

Introduction to FEMdesigner AD: Heat Stress Tutorial

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.

The geometry length units are inches are automatically the same as those the drawing and these correspond to pound-force units. A metric drawing would use SI units. The mouse operations for view rotation and panning are the same as in Alibre. In the current version of Alibre the Femdesigner tab is available from the main menu and consists of the button bar shown below.

Press 'Activate Menu' to unhide the buttons. If you need to subsequently alter the model you should first deactivate the menu as Alibre may produce an error message otherwise.
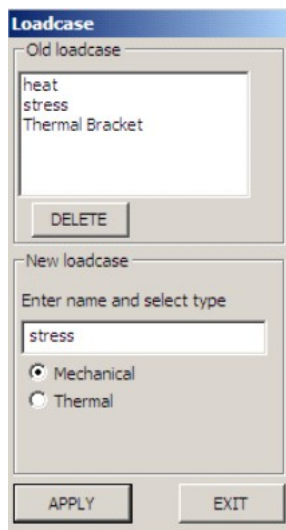
Open File / Create Loadcase

The corresponding zip file has an Alibre Design part file; Thermal Bracket_AD_PRT already referenced and opened in the Steady State Heat Transfer Tutorial.

• Click on 'Loadcase' command button to bring up the 'Loadcase' dialog box.

• Select 'Create New Loadcase', provide a name, click 'Mechanical' and click 'Apply'.

• Note that we should be using the same mesh as in the Steady State Heat Transfer Tutorial so no need to rebuild it.
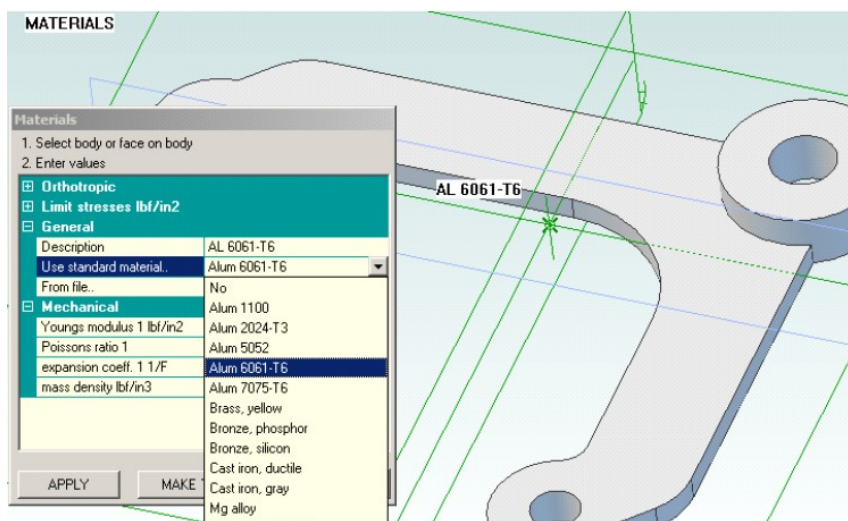
Assign Materials

• Click on 'Attach Materials' command.

• Select any face.

• Select the same material from the 'Standard Materials' drop-down list as in the Steady State Heat Transfer Tutorial.

• Click 'Apply'.
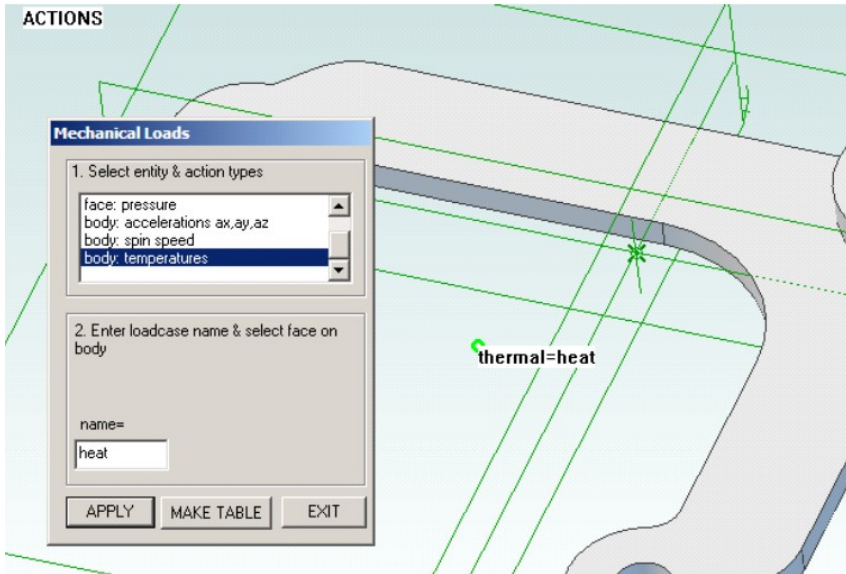
Apply Temperatures

- Select the top exposed face of the centre rod.
- Click on 'Plot/Add Actions' command.
- Select 'Body temperatures'.
- Enter the name of the thermal loadcase you previously created in the Steady State Heat Transfer Tutorial, in this case, it is "heat".
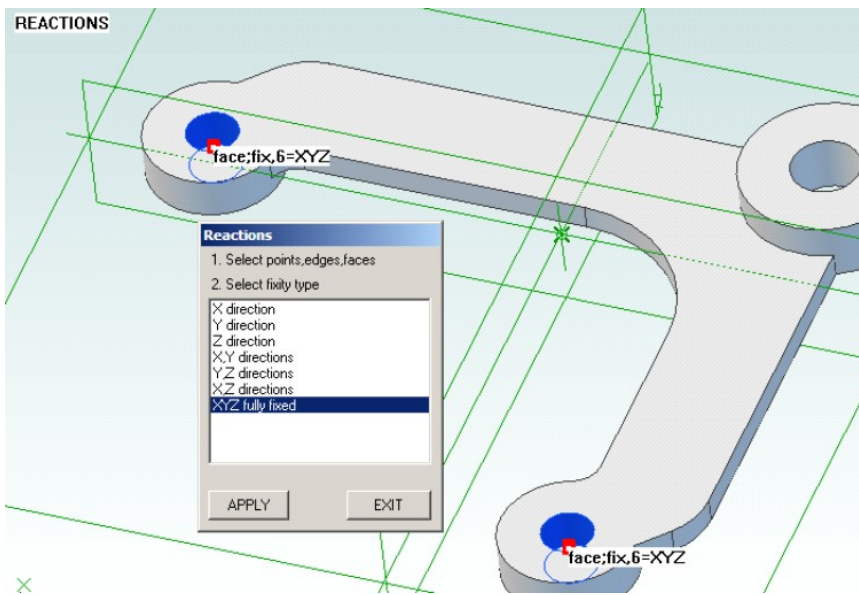
Fix Mounting Bores

To simulate the bracket being mounted on two fixed shafts:
- Click on 'Plot/Add Reactions' command.
- Select "XYZ fully fixed".
- 'Shift-select' both interior bores at the ends of the arms of the bracket, as shown.
- Click 'Apply'.
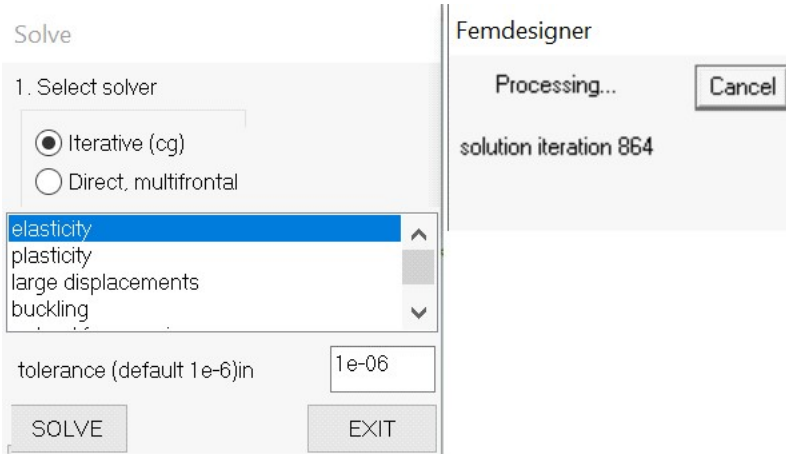
Solving

Now that all boundary conditions are correctly applied, the FE model solution can be started.

- Click 'Solve' command.
- Select 'Iterative' and 'elasticity' on dialog box.
- Click on 'SOLVE'. A status box with solver progress information appears.
- When the status box disappears, the solution is ready.
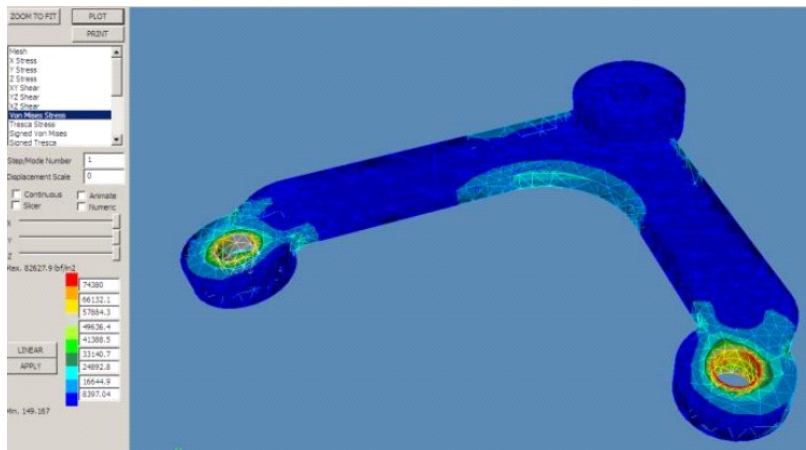
View Results

Click the 'Plot Results' command to bring up the results dialog box.

- Select 'Von Mises Stress' from the list.
- Plot the XYZ displacement and enter a number in the 'Displacement Scale' box to multiply the actual displacement shape.
- Animate the deflection by checking the 'Animate' box.

You could add other 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.

Note that if stresses are zero you could try adding the suffix .out to the thermal loadcase name.