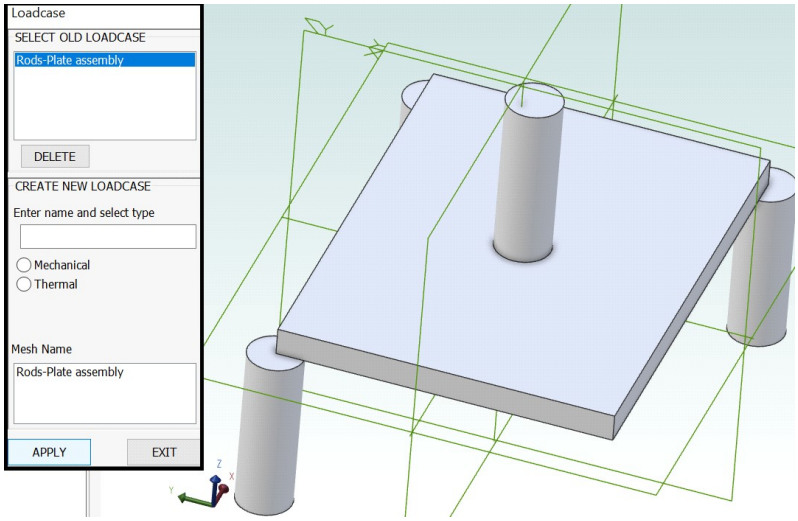
# **FEMdesigner AD:** Assembly Contact Analysis Tutorial

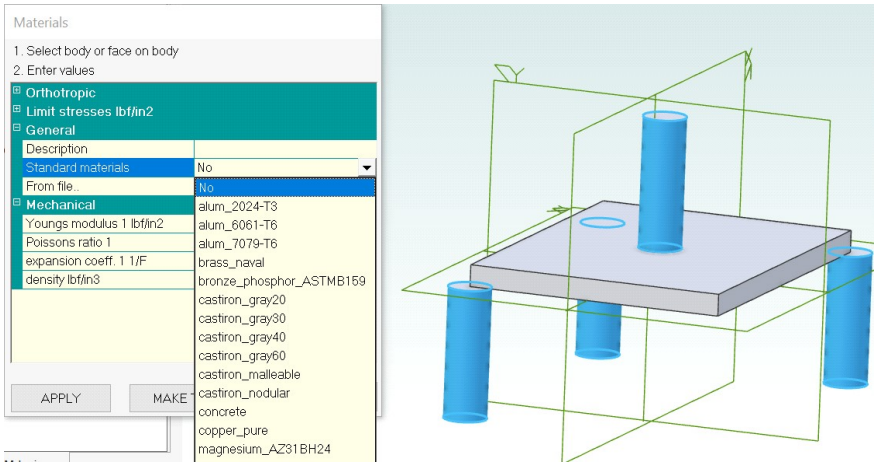
In this tutorial, you will open an existing Alibre Design assembly file, assign materials to the different parts in the assembly, create a mesh of all parts, apply a load to one part, apply boundary conditions to the supporting parts, specify part-to-part interfaces, solve and display results. Refer to the 'activated' command bar below.

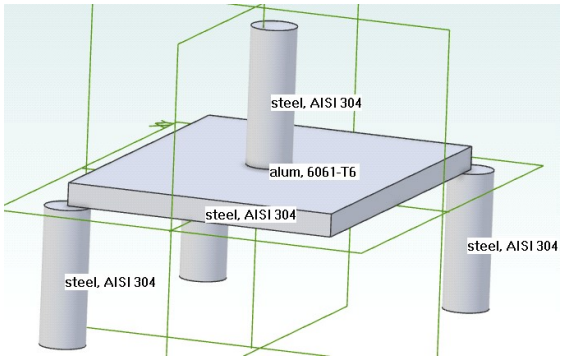




**Open the Assembly**

The corresponding zip file has an Alibre Design assembly file and its associated part files; assembly.ad_asm, Rod.ad_prt and Plate.ad_part.

- Unzip the files to a folder.
- Click on 'Loadcase' command button to bring up the 'Loadcase' dialog box.
- Select 'Create New Loadcase' and 'Mechanical' or just 'Old Loadcase' (if it exists) and click 'Apply'.

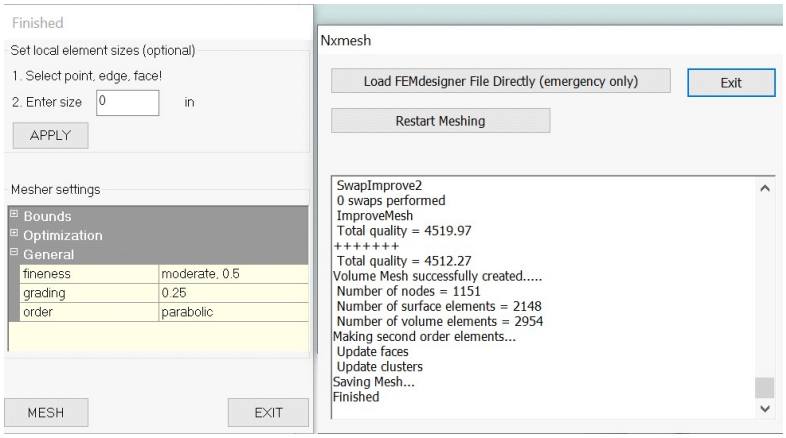


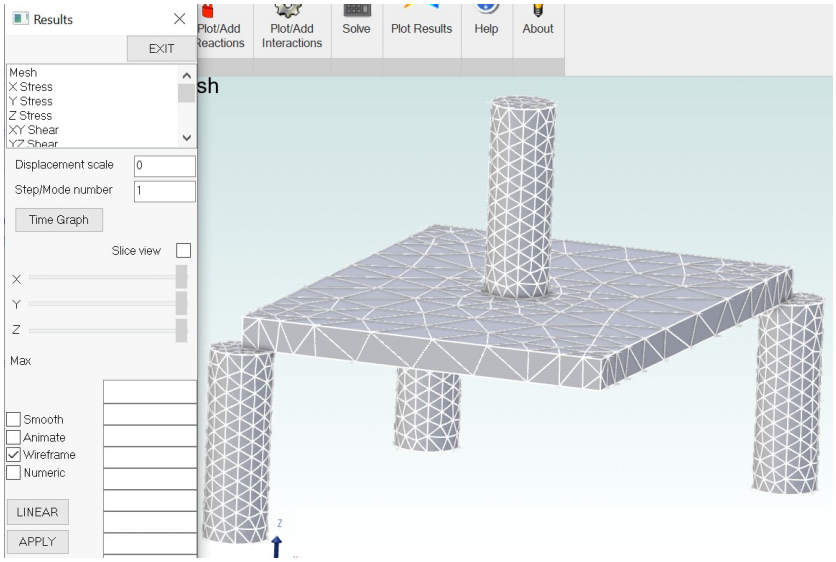




**Assign Materials**

- Click on 'Attach Materials' command.
- Select a face on each of the four rods using the 'shift-select' method in Alibre.
- Select a material (eg steel 304) from the 'Standard Materials' drop-down list and click 'Apply'
- Select a face on the plate and attach a different material (eg alum 6061) and click 'Apply'.
- All components should now have a material displayed, as shown.

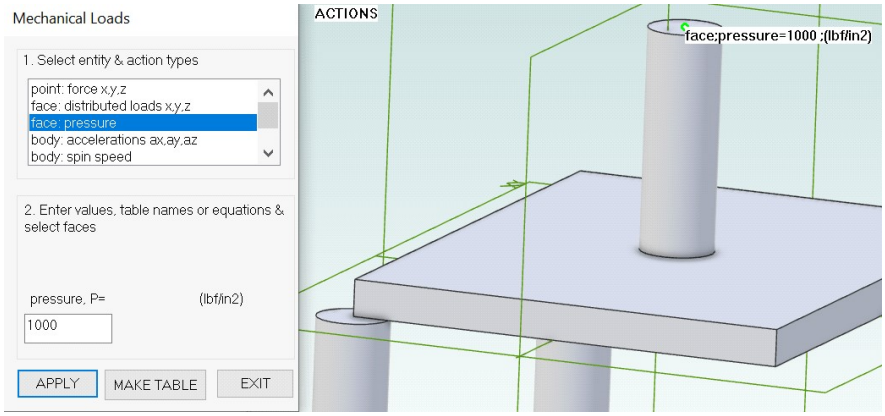
- Click ‘Exit’.




### Create/View Mesh

- Click on ‘Build Mesh’ command to display the assembly mesh dialog box. Mesh properties can be adjusted here but for now just use the default parameters.
- Click the ‘MESH’ button. The mesher will automatically mesh all parts in the assembly in a separate dialog box and display “Finished” when complete.
- ‘EXIT’ both mesh dialog boxes.
- Click on the “Plot Results” command and the Results window appears with the mesh automatically shown. Highlight with the ‘Wireframe’ option.
- Click ‘EXIT’.

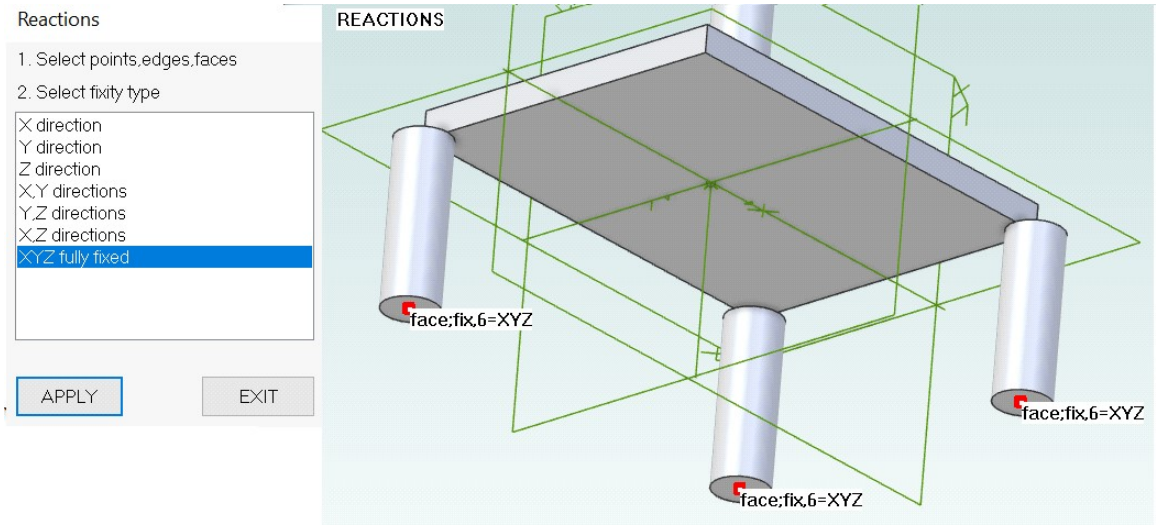



### Apply Loads

- Select the top exposed face of the centre rod.
- Click on ‘Plot/Add Actions’ command.
- Select ‘Face pressure’ and enter 1000 in the dialog.
- Click ‘APPLY’ to see the pressure overlaid on the model.
- Click ‘EXIT’.
- Rotate the assembly to show the bottom faces of the 3 support rods.

**Fix Supports**

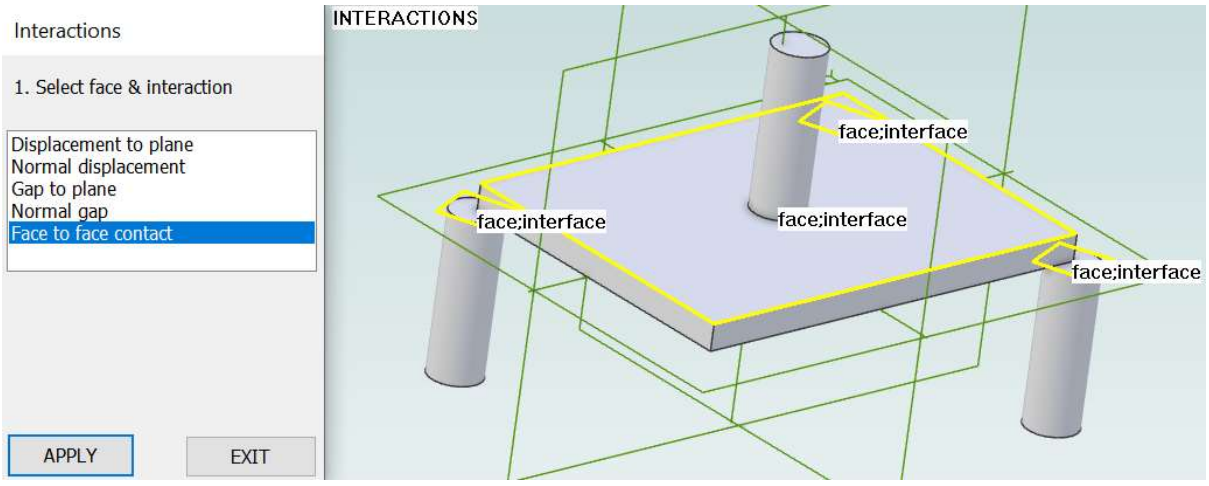
- Click 'Plot/Add Reactions' command.
- Select the 3 exposed rod end faces using 'shift-select'.
- Select "XYZ fully fixed" then 'APPLY'.
- Reactions will be shown overlaid on the model.
- Click "EXIT".

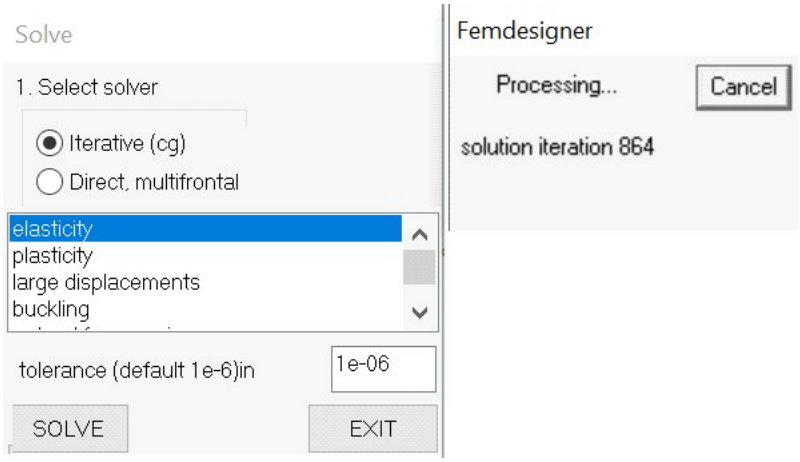



**Specify Contact Faces**

When two opposing faces contact and transfer loads, only one of the two face contacts need be specified. There is a loaded rod pushing against a plate, which pushes against its supporting rods, so the interfaces will be between the top face of the plate and the top faces of the supporting rods

- Click on the 'Plot/Add Interactions' command.
- Click on the 'Face to face contact' option on the dialog box.
- Select the top circular faces of the supporting rods and the large exposed face of the plate using 'shift-select'.
- Click on "APPLY". Interfaces will be displayed graphically as shown below.
- Click "EXIT".

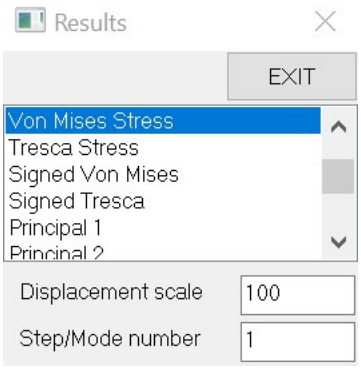





**Solving**

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

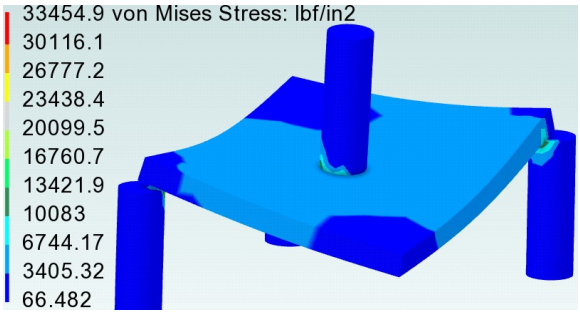
- Click 'Solve' command.
- Select 'Iterative' and 'elasticity' on dialog box.
- Click on 'SOLVE'. A status box with solver progress information appears.
- When the status box disappears, the solution is ready.

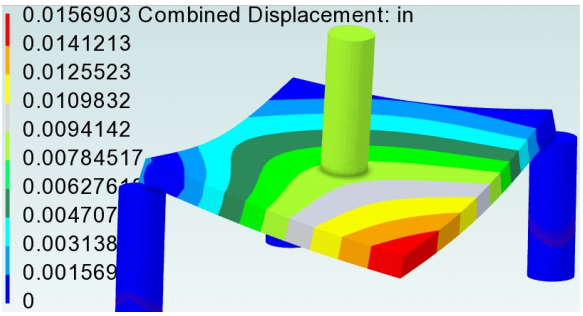



**View Results**

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

- Select 'Von Mises Stress' from the list of available results.
- Rotate the model to see areas of stress concentrations, noticing that these areas are where expected; on the edges of the support rods where the plate is in contact and the edge of the loaded rod where the plate is most strongly supported.
- Plot other types of stress that are available, as desired.
- To better visualize how the parts react to each other, plot the displacement and enter 100 in the 'Displacement Scale' box to multiply the actual displacement shape by 100 times.
- Animate the deflection by checking the 'Animate' box.




If you don't achieve similar results to those above then perhaps on your computer extra reactions are required on the centre of the plate and/or on the top of the centre rod to avoid them sliding away along the frictionless planes. On other models, take advantage of planes of symmetry by removing symmetric parts of the model and replacing the missing parts with 'Reactions'.