Introduction to FEMdesigner AD: Assembly Contact Analysis Tutorial

FemforAD has a menu structure that is logically organized to represent the step-by-step process of creating an analysis. These steps are:

1. Create Geometry in Alibre. Parts and assemblies can be analysed with the latter connected by frictionless contact.
2. Assign Materials. A selection of materials is provided but their properties can be overridden to create your own material.
3. Create Boundary Conditions. "Boundary conditions" refers to the actions (loads), reactions (restraints) and other conditions that will govern the behaviour of the geometry.
4. Create 'Mesh'; a collection of individual volumetric finite 'elements' connected together at 'node' points that fill in the 3D volume of the CAD model.
5. Solve; a system of equations is generated and solved through numerical analysis to a given error tolerance. The user does not need to know the mathematics involved in this process.
6. Plot Results. Once the equations are solved, the results are viewed as contour plots. These include stresses, displacements, temperatures, plastic strains, natural frequencies, etc.

In this tutorial, we will perform a linear elastic analysis which involves.

- Create a loadcase for the analysis. Several loadcases can be created with different loads and analysis types.
- Assign materials to the parts.
- Create finite element mesh.
  - Automesh using defaults.
  - View the mesh.
- Apply boundary conditions.
  - Actions: Pressure.
  - Reactions: Fixed ends.
  - Interactions: Face to face contact.
- Solve for resultant elastic stresses
- Visualize the results Refine mesh, solve and visualize.

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.
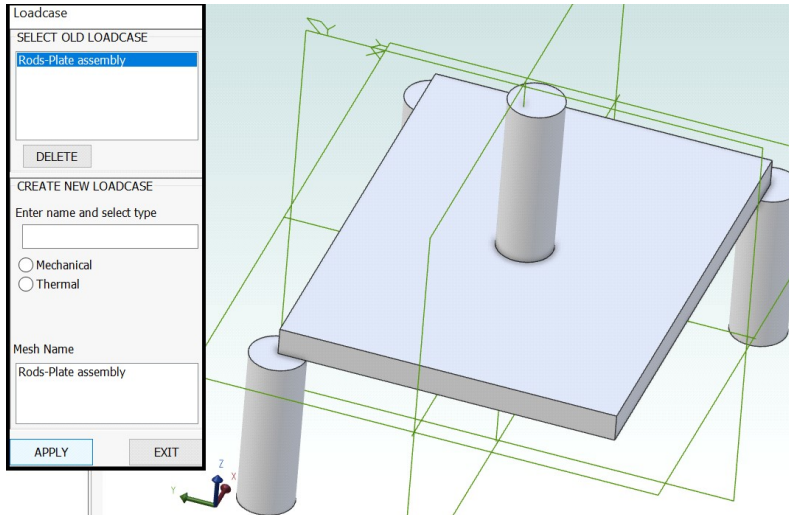
Open File / Create Loadcase

The corresponding zip file has an Alibre Design 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', type a name and click 'Mechanical' or just 'Old Loadcase' (if it exists) and click 'Apply'.
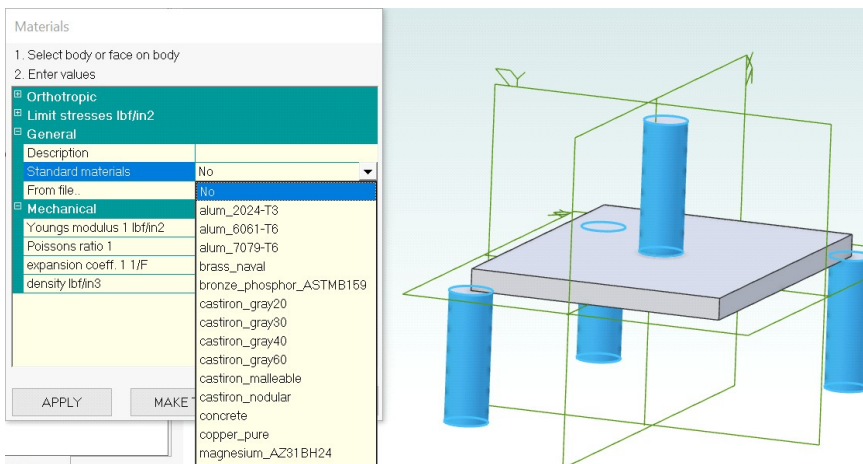

Assign Materials

• 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 new material (eg alum 6061). Click 'Apply'.
• All components should now have a material displayed, as shown.
• Click 'Exit'.
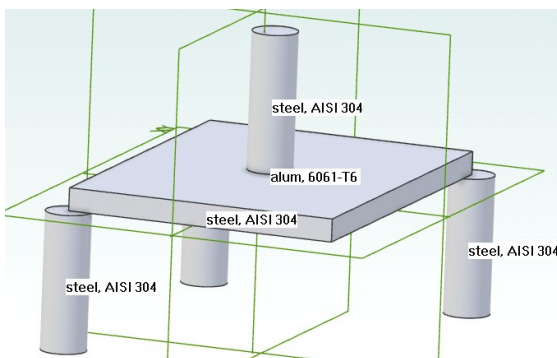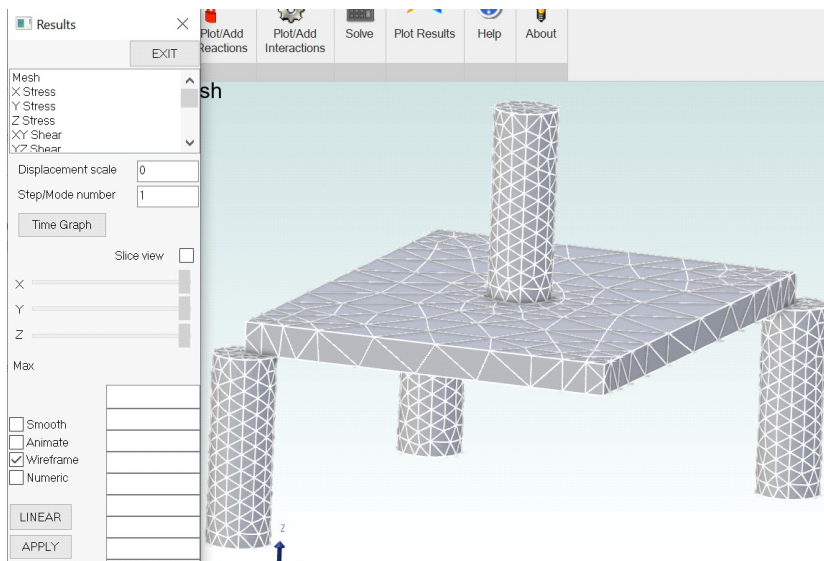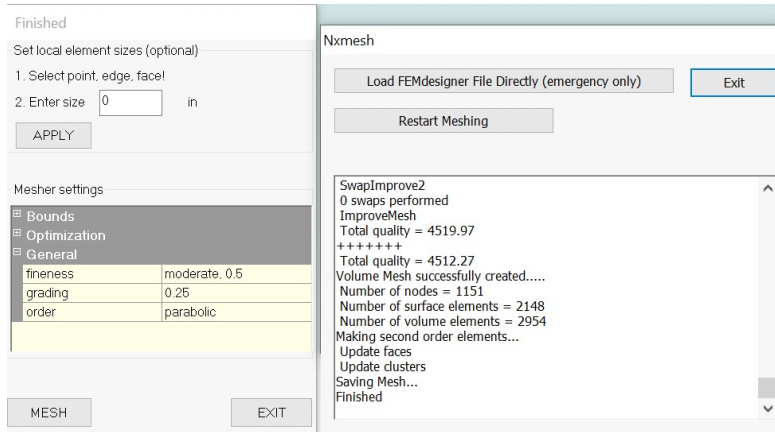
Create/View Mesh

• 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'.
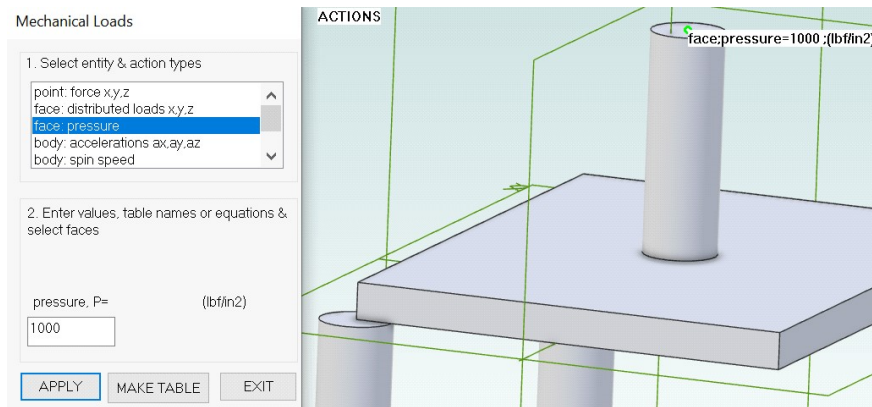
**Apply Loads**

• 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".
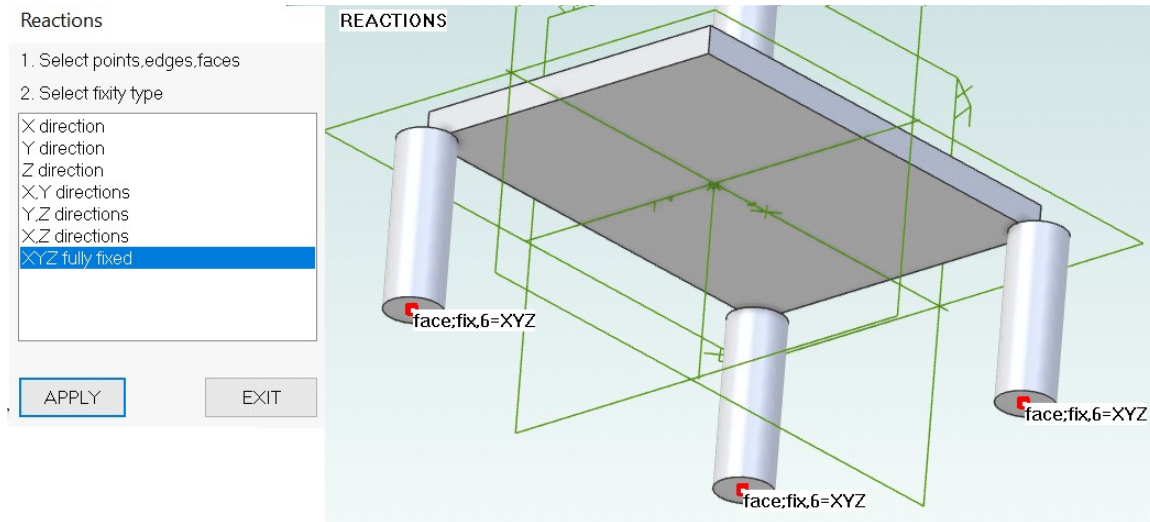
Specify Contact Faces

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.
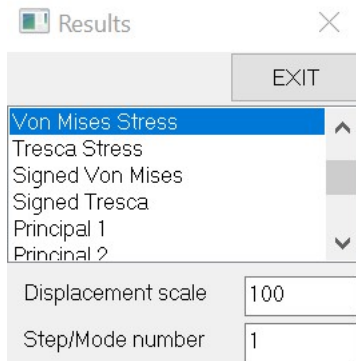- Click "EXIT".
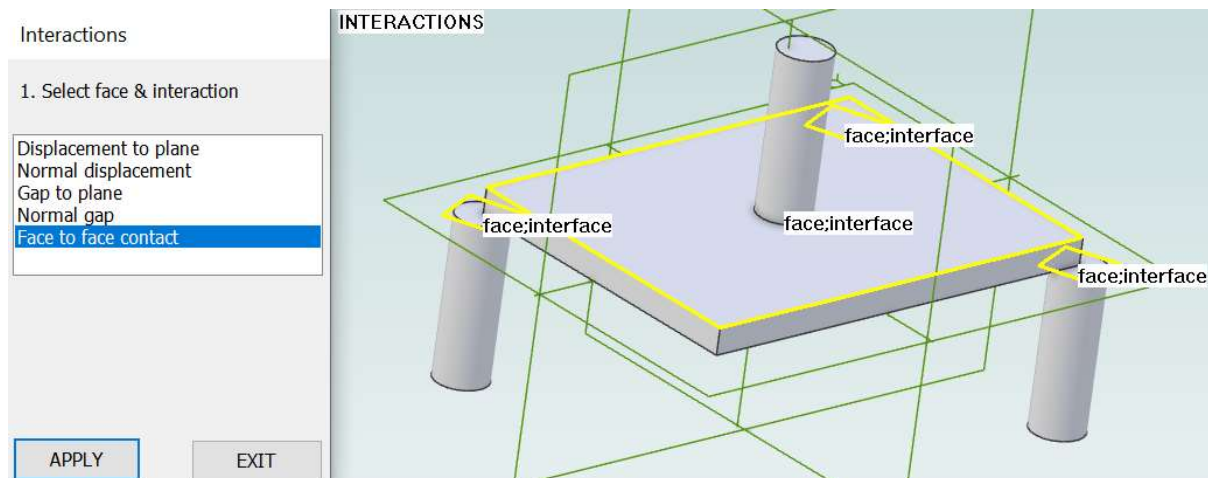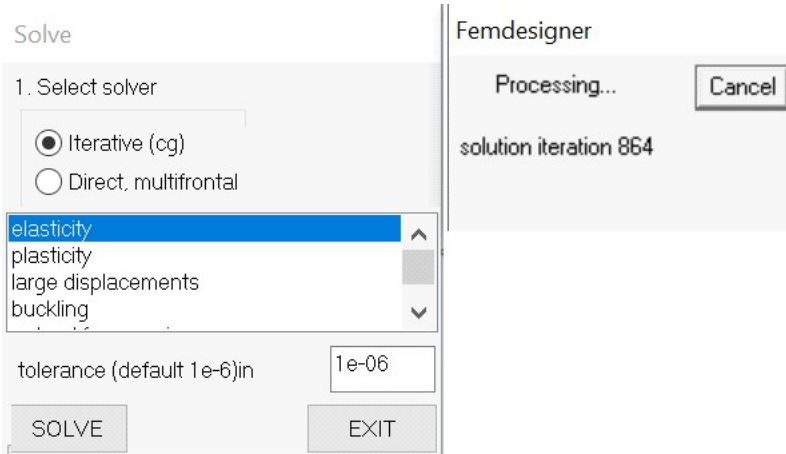
Solving

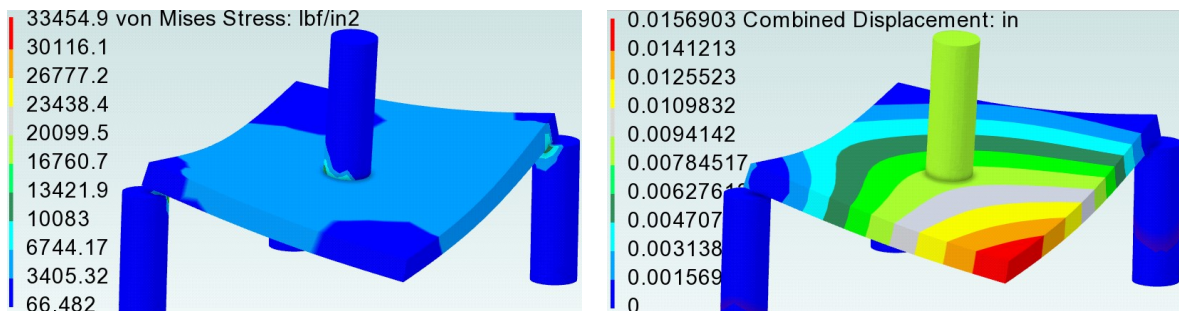Now that all boundary conditions are correctly applied, the FE model solution can be started.

- 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.

Solve

1. Select solver
   - ⦿ Iterative (cg)
   - ○ Direct, multifrontal

  elasticity
  plasticity
  large displacements
  buckling

  tolerance (default 1e-6)in   1e-06

  SOLVE   EXIT

Femdesigner

Processing...   Cancel

solution iteration 864

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.

Results   EXIT

Von Mises Stress
Tresca Stress
Signed Von Mises
Signed Tresca
Principal 1
Principal 2

Displacement scale   100

Step/Mode number   1

33454.9 von Mises Stress: lbf/in2
30116.1
26777.2
23438.4
20099.5
16760.7
13421.9
10083
6744.17
3405.32
66.482

0.0156903 Combined Displacement: in
0.0141213
0.0125523
0.0109832
0.0094142
0.00784517
0.00627616
0.004707
0.003138
0.001569
0

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'.