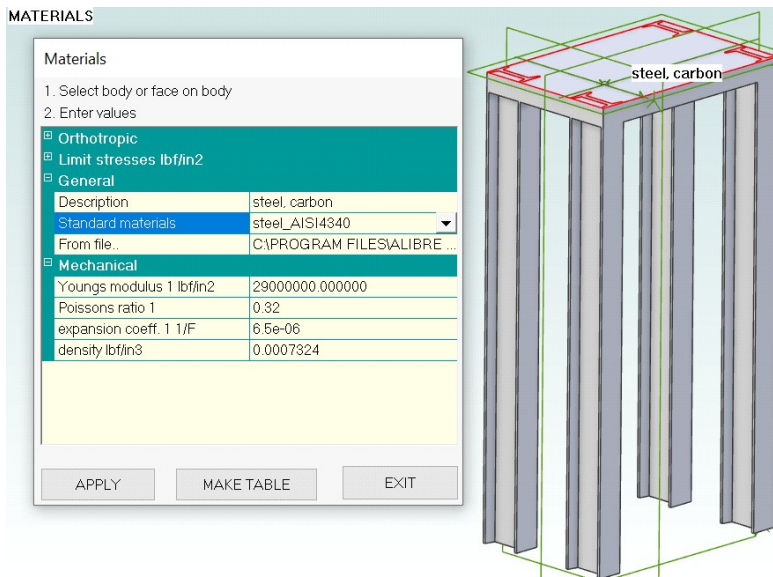


## FEMdesigner AD: Buckling Tutorial

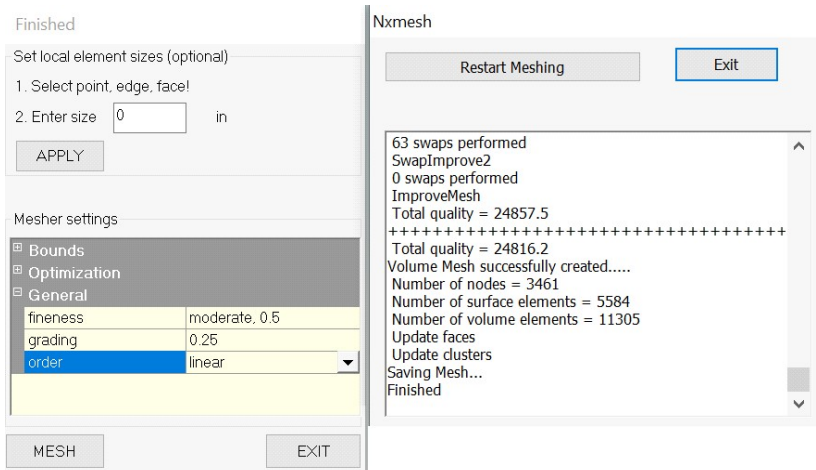
In this buckling tutorial you will calculate the buckling modes and corresponding load factors of a series of four I-beams supporting a plate, forming a rather tall, slender, table-like structure. Refer to the 'activated' command bar below.



### Open File and Assign Material

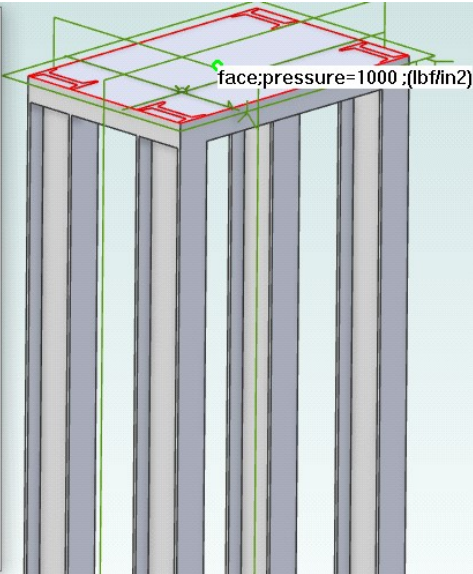
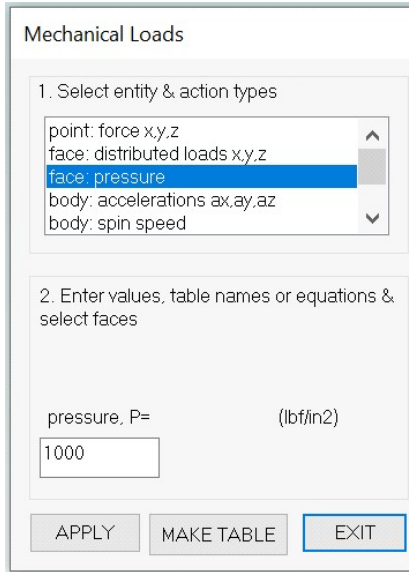
The corresponding zip file has an Alibre Design part file; I beam table.ad\_prt. Unzip it to a suitable folder.

- Click on 'Loadcase' command button. Select 'Create New Loadcase', click 'Apply' and 'Exit'.
- Click on 'Attach Materials' command and select a face.
- Select a material from the drop-down list and click 'Apply'. The material will be displayed.
- Click 'Exit'.



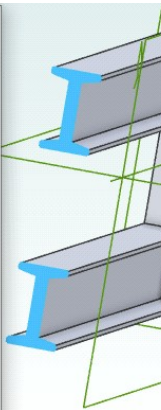
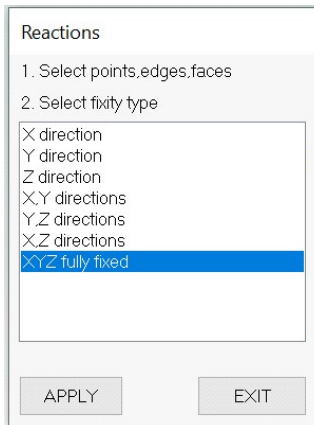
### Create/View Mesh

- Click on 'Build Mesh' command.
- Change the 'order' to 'Linear'. Leave other settings at their default.
- Click the 'MESH' button to bring up the progress dialog box which displays "Finished" when done.
- 'EXIT' both mesh dialog boxes.



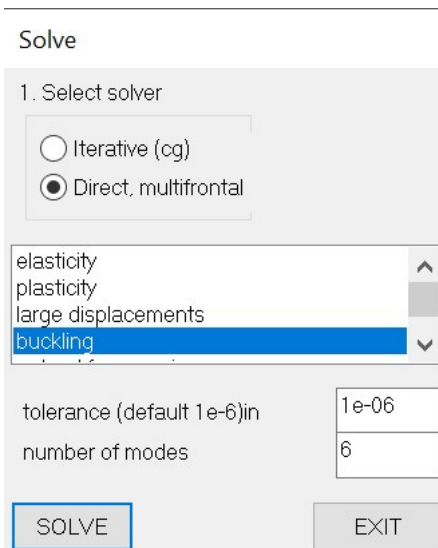
### Apply Load

- Select the top face of the table.
- Click on 'Plot/Add Actions' command.
- Select 'Face pressure' and enter 1000 in the dialog.



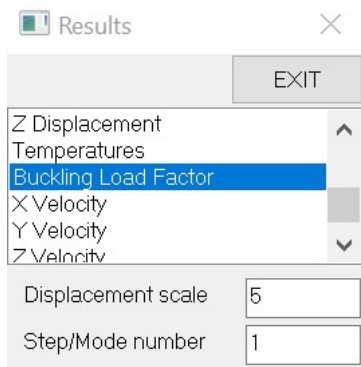
### Fix Base

- Click 'Plot/Add Reactions' command.
- Restrain the bottom faces of the I-beam legs as shown.
- Select 'XYZ fully fixed', click 'APPLY' then click 'EXIT'.



### Solve for Buckling

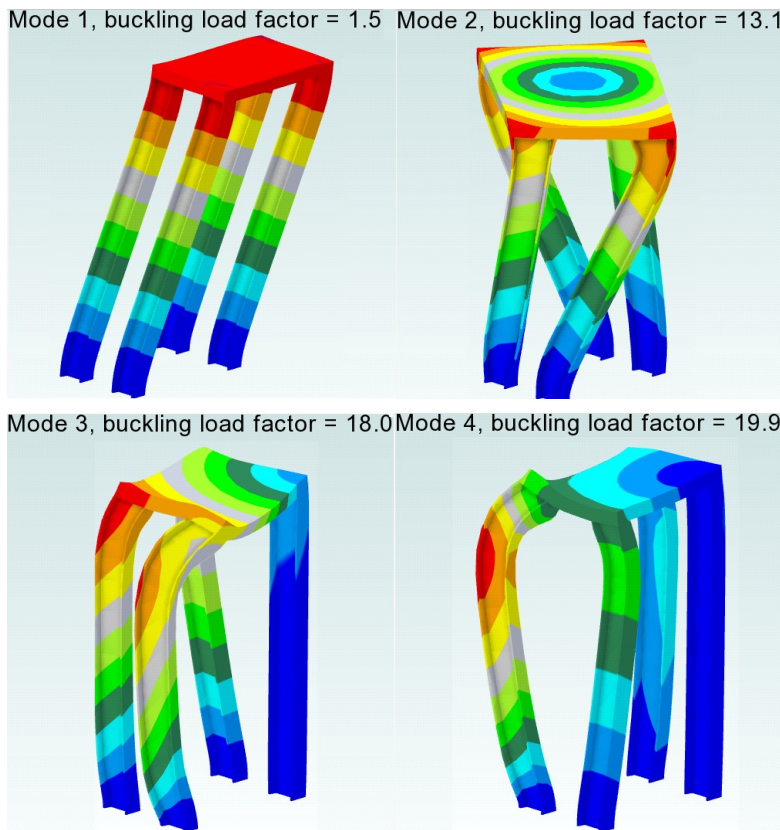
- Click 'Solve' command.
- Select 'Direct' and 'buckling' on dialog box.
- Type 6 in the 'number of modes' box. Only 1 mode (the lowest) is required but the other modes can be instructive.
- Click on 'SOLVE'. A status box with solver progress information appears.
- When the status box disappears, the solution is ready.



**View Results**

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

- Enter the number 5 in the 'Displacement scale' box and select 'Buckling Load' from the results list.
- The load factor for each buckling mode is displayed on screen.
- Click the 'Animate', checkbox to see the buckling in motion. Unclick it to stop the animation.
- Enter 2 in the "Step/Mode number" box and re-click on the 'Buckling Load' selection. Repeat for modes 3 to 6.



The first 4 modes are shown on the left.

The buckling factor is the same as a safety factor so if the load is multiplied by that value then buckling of the geometry is predicted. The minimum safety/loading factor (to account for manufacturing/shape imperfections etc) for ASME pressure vessel codes is 2.5 but this can vary between 1.5 and 5 in different design codes (aerospace, structural etc). You must check this! European/other design codes have similar minimum safety/loading factors to the US codes.

The numbers on the contour plot are meaningless and are only used to show the shape, which comes from the normalised 'Eigenvector'. The buckling load is the corresponding 'Eigenvalue'.

Note that the mesh was made 'Linear' rather than 'Parabolic' because stresses are not required so we can use a coarser mesh. This computationally helps offset the fact that for buckling we usually need to mesh the entire body because modes are not always symmetric (as demonstrated above) even when the geometry is symmetric. Refine the mesh to ensure you have a minimum value which is mesh independent.