



# 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.





Finished		Nxmesh
-Set local element sizes 1. Select point, edge, f	(optional)	Restart Meshing Exit
2. Enter size 0 in APPLY		63 swaps performed SwapImprove2 0 swaps performed ImproveMesh Total quality = 24857.5
<sup>⊞</sup> Bounds <sup>⊞</sup> Optimization <sup>⊟</sup> General		Total quality = 24816.2 Volume Mesh successfully created Number of nodes = 3461 Number of surface elements = 5584
fineness grading order	moderate, 0.5 0.25 linear	Number of soluce elements = J364    Number of volume elements = 11305    Update faces    Update dusters    Saving Mesh    Finished
MESH	EXIT	

## **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.



Mechanical Loads	
1. Select entity & action types point: force x.y.z face: distributed loads x.y.z face: pressure body: accelerations ax.ay.az body: spin speed	face;pressure=1000 :(lbf/in2)
2. Enter values, table names or equations & select faces	
pressure, P=  (lbf/in2)    1000	

### **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

1. Select solver	
<ul><li>Iterative (cg)</li><li>Direct, multifrontal</li></ul>	
elasticity plasticity large displacements buckling	
tolerance (default 1e-6)in number of modes	1e-06 6
SOLVE	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.



© i	Convright	FFMdesig	ner	Itd.
	copyright	FLIVIUESIg	;iici	Luu.

🔳 Results	$\times$	
	EXIT	
Z Displacement Temperatures	^	
Buckling Load Factor X Velocity		
Y Velocity Z Velocity	~	
Displacement scale	5	
Step/Mode number	1	

### 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.

Mode 1, buckling load factor = 1.5 Mode 2, buckling load factor = 13.1



The first 4 modes are shown on the left.

Mode 3, buckling load factor = 18.0 Mode 4, buckling load factor = 19.9





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.