

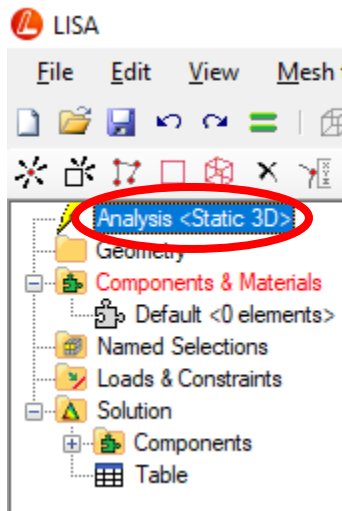
## Using LISA FEA for Modal Analysis of DML Speaker Panel

### Summary

This describes the use of LISA to estimate the mode shapes and natural frequencies of a 400x600 mm x 25mm thick EPS panel. For All entries use

### Setup

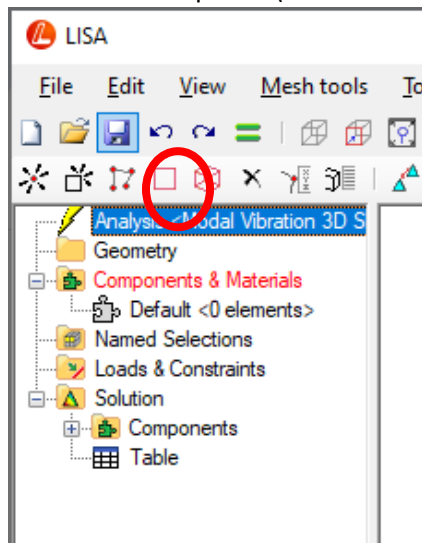
1. Open LISA
2. Right click on "Analysis<3DStatic>" and select EDIT



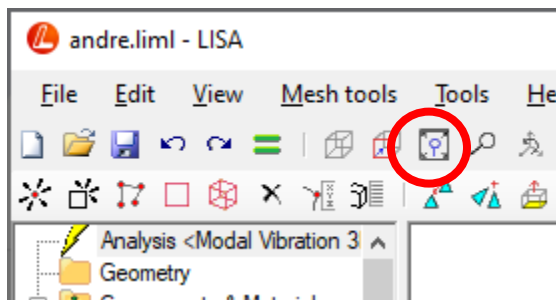
3. Select 3D
4. Select Analysis: Modal Vibration 3D Shell and Beam
5. Select number of modes: Choose 50 (fewer will run faster), then select "OK"

### Create Plate Model

1. Select "Quick Square" (This creates a 1 meter x 1 meter square plate).



2. From top menu select Mesh Tools/Refine/Custom
3. Set R=48, S=32 and click “Apply”, then “Close” (This creates a 48x32 mesh within the square).  
*Note, if using the Free Demo version, set R=24, S=16.*
4. The four corner points of the mesh will appear blue. Click in the area near the blue points but outside the meshed area to deselect the four corner points. They should turn red (deselected) when you click.
5. Select Mesh Tools/Change Element Shape
6. Select “Quad8” (it will likely already be selected by default) and click “OK”. This will add four mid-side nodes to each element. It will take a few seconds to happen, so just wait for it.
7. Select Mesh Tools/Scale
8. Enter Scale factor X=0.6, Enter Scale Y Factor = 0.4 and click OK. (This changes the plate size to 600 mm x 400m)
9. Click the “Fit to Screen” tool to resize the display.



### Input Material Properties

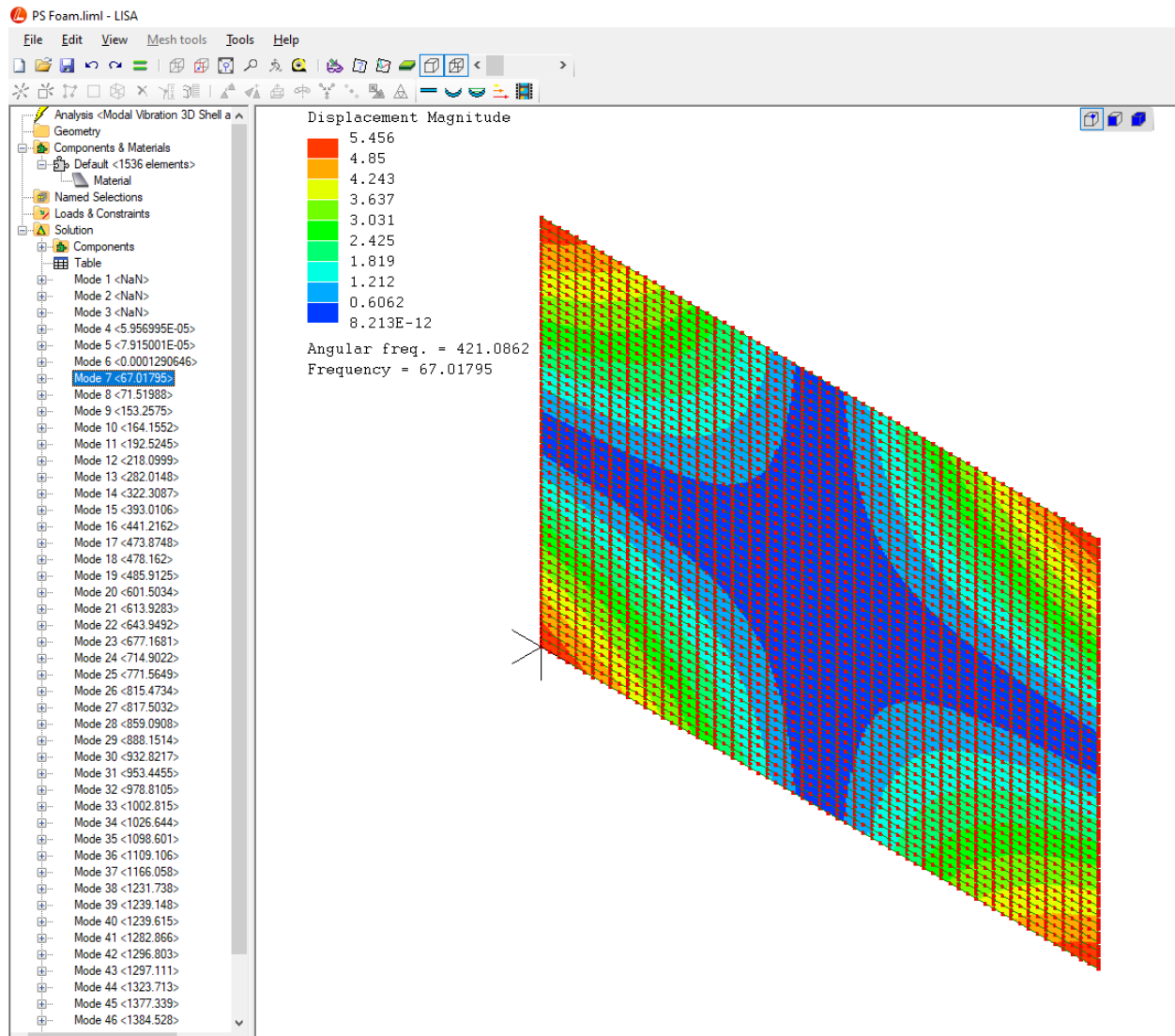
1. Right Click on “Default<1536 elements>” and select “Assign New Material”
2. Select “Plate/shell/membrane”, then enter thickness = .025, but don’t click “close” just yet. This chooses the plate thickness to be 25 mm.
3. Select the “Mechanical” tab, and click “Isotropic”, make entries as shown below and then click “close”
  - a. Young’s modulus =  $4e7$  (i.e. 40 MPa)
  - b. Poisson’s Ratio = 0.25
  - c. Density = 40 (i.e. 40 kg/m<sup>3</sup>)

### Save and Run Model

1. Click “Save” icon and name your file.
2. Click the green equal sign to run the model.
3. The blue “solver” box will appear. The solver will run for a few seconds and when it’s done the box will turn green.

### Review Results

1. On the left side a list of 50 Modes should have appeared when the solver finished.
2. Ignore the Modes 1-6. These are the so called “rigid body motions” and not real modes.
3. The remaining list (Modes 7 to 50) are the actual natural frequencies. Click on any one of them and the mode shape corresponding to the mode that you selected will be displayed.



### Adding Perimeter Constraint (so called “simple” support around the entire perimeter)

1. Select XY View
2. Select all the nodes around the perimeter: Hold down “ctrl” and use the cursor to select all the nodes on each of the four sides. The selected nodes should turn from red to blue.
3. Right click on any of the highlighted nodes, then select “Loads and Displacements” and “New Displacement”
4. Select “Z”, and click “OK”.
5. Save and Run.

**Bonus Hint: Eliminate the rigid body modes from the solutions**

1. On the left side, under “Named Selection” click on “Unnamed<x nodes>”. This reselects the same nodes you selected in the previous section.
2. Repeat steps 3-5 above but instead of selecting “Z”, select “X”.
3. Repeat again and select “Y” in addition.
4. Save and run.
- 5.

**Fully Clamped Constraints**

6. On the left side, under “Named Selection” click on “Unnamed<x nodes>”. This reselects the same nodes you selected in the previous section.
7. Right click on any of the highlighted nodes, then select “Loads and Displacements” and “New Displacement” and “On Selected Nodes” and then “new rotx”
8. Enter “0” and “OK”
9. Repeat steps 2 and 3 but choosing “new roty” and setting that to “0”
10. Save and Run.