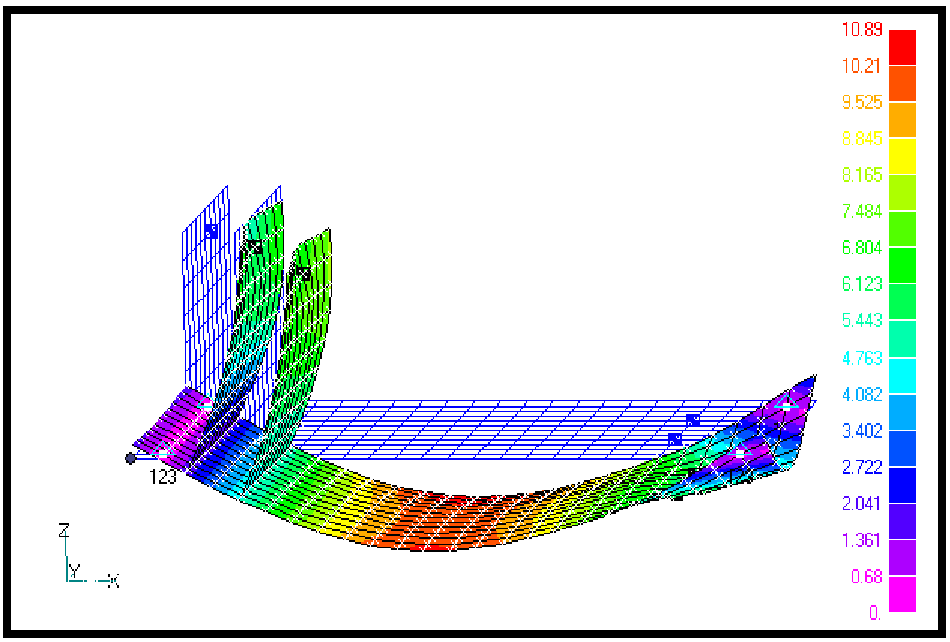
WORKSHOP 12a

Modal Analysis of a Printed Circuit Board



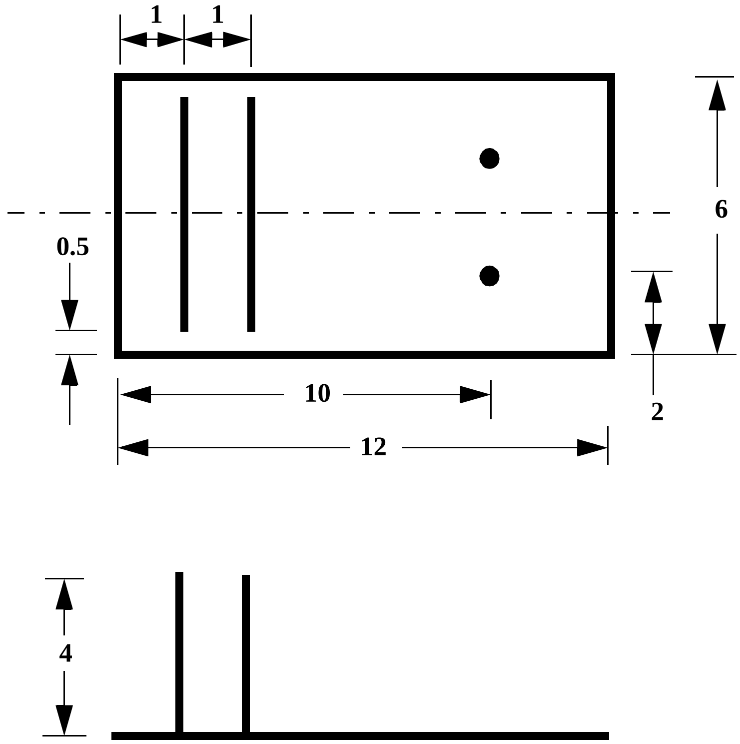
Objectives:

- Create the finite element model using the given dimensions
- Input the boundary conditions
- Create equations for relative motion between selected points
- Run a modal analysis saving a database for susequent analysis
- View the results

Model Description:

Create a finite element model of a simple electronic assembly using plate elements and lumped masses. The plate elements will include a none structureal mass to account for the distributed mass of its components and lumped mass to represent selected items. Write constraint equations for relative motion between selected points. Then run an analysis to calculate normal modes and save the solver database for use in subsequent analysis.

Figure 12a.1 - Printed Circuit Board Dimensions



A printed circuit board is pinned at its corners. The material properties data are given below. Determine the first 10 modal frequencies of the circuit board.

Circuit Board Properties:	
Elastic Modulus, E =	3.3E6 psi
Poisson's Ratio, ν =	0.3
Mass Density =	0.1 lbs./in^3
Thickness, t =	0.125 in.
Capacitor Properties:	
Mass, M =	1 lb
Ix =	4 lbs./in^2
Iy =	4 lbs./in^2
Iz =	3 lbs./in^2
Offset from Board =	2
CPU Properties:	
Mass, M =	0.25 lb
Ix =	4 lbs./in^2
Iy =	4 lbs./in^2
Iz =	3 lbs./in^2

Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0 and begin to create a new model.

   Double click on the icon labeled **MSC.Nastran for Windows V4.0.**

   On the *Open Model File* form, select **New Model.**

   Open Model File: **New Model**

2. Create the geometry of the main circuit board.

   Geometry/Surface/Corners...

X:	**0**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**12**	*Y:*	**0**	*Z:*	**0**	**OK**
X:	**12**	*Y:*	**6**	*Z:*	**0**	**OK**
X:	**0**	*Y:*	**6**	*Z:*	**0**	**OK**

   Cancel

   Turn off the workplane.

   Tools/Workplane... (F2)

   ❑ **Draw Workplane**

   Done

   View/Autoscale (Ctrl+A)

3. Create two curves that represents the base of the two daughter cards.

Geometry/Curve - Line/Project Points...

X:	**1**	*Y:*	**0.5**	*Z:*	**0**	**OK**
X:	**1**	*Y:*	**5.5**	*Z:*	**0**	**OK**
X:	**2**	*Y:*	**0.5**	*Z:*	**0**	**OK**
X:	**2**	*Y:*	**5.5**	*Z:*	**0**	**OK**

Cancel

4. Create the two daughter cards on the main circuit board

Geometry/Surface/Extrude...

<select the 2 curves on the surface - ID # 9,10>

OK

Base X:	**0**	*Y:*	**0**	*Z:*	**0**
Tip X:	**0**	*Y:*	**0**	*Z:*	**4**

OK

Cancel

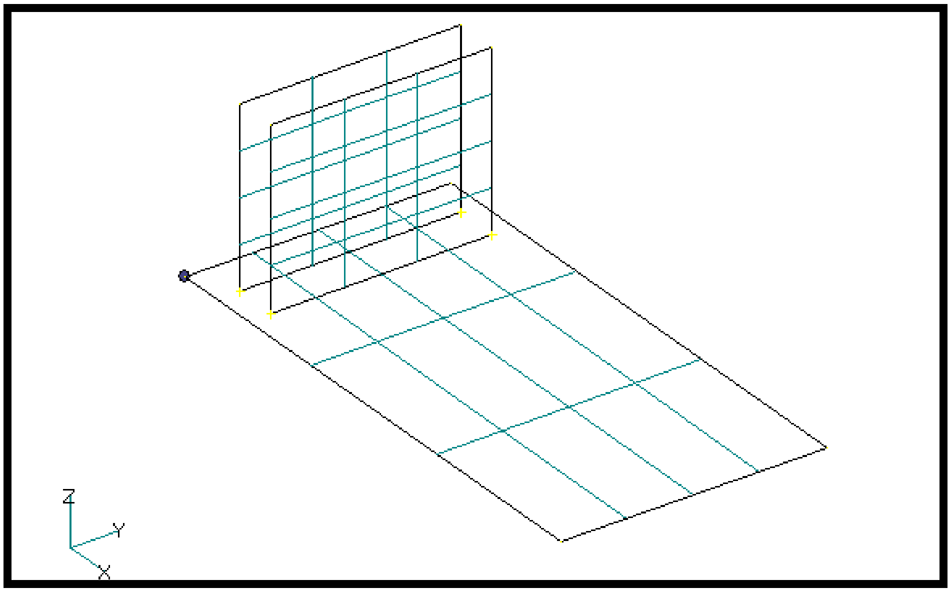
View/Rotate... (F8)

Dimetric

OK

Your display should look like the following. Autoscale if necessary.

Figure 12a.2 - Geometry of Circuit Board



5. Create the materials for the model.

Model/Material...

Title:	**fiberglass**
Youngs Modulus:	**3.3E6**
Poisson's Ratio:	**0.3**
Mass Density:	**0.1**

OK

Cancel

6. Create a properties for the model. The plate property includes the non-structural mass term.

From the pulldown menu, select **Model/Property**.

Model/Property...

Title:	**circuit_board**
Thickness, Tavg or T1:	**0.125**

To select the material, click on the list icon next to the databox and select **fiberglass.**

Material:	**1.. fiberglass**
Nonstructural area/mass:	**0.01**

OK

Title:	**capacitor**

Elem/Property Type...

Other Elements:	● **Mass**

OK

Mass, M or Mx:	**1**
Inertia, Ixx:	**4**
Iyy:	**4**
Izz:	**3**
Offset from Node: Z:	**2**

OK

Title:	**cpu**
Mass, M or Mx:	**1/16**
Inertia, Ixx:	**0**
Iyy:	**0**
Izz:	**0**

OK

Cancel

7. Create the finite element mesh.

Mesh/Mesh Control/Default Size...

Size:	**0.5**
OK	

Mesh/Geometry/Surface...

Select All	
OK	
Property:	**1.. circuit_board**
OK	

8. Change the view of the model to hidden line and geometry off.

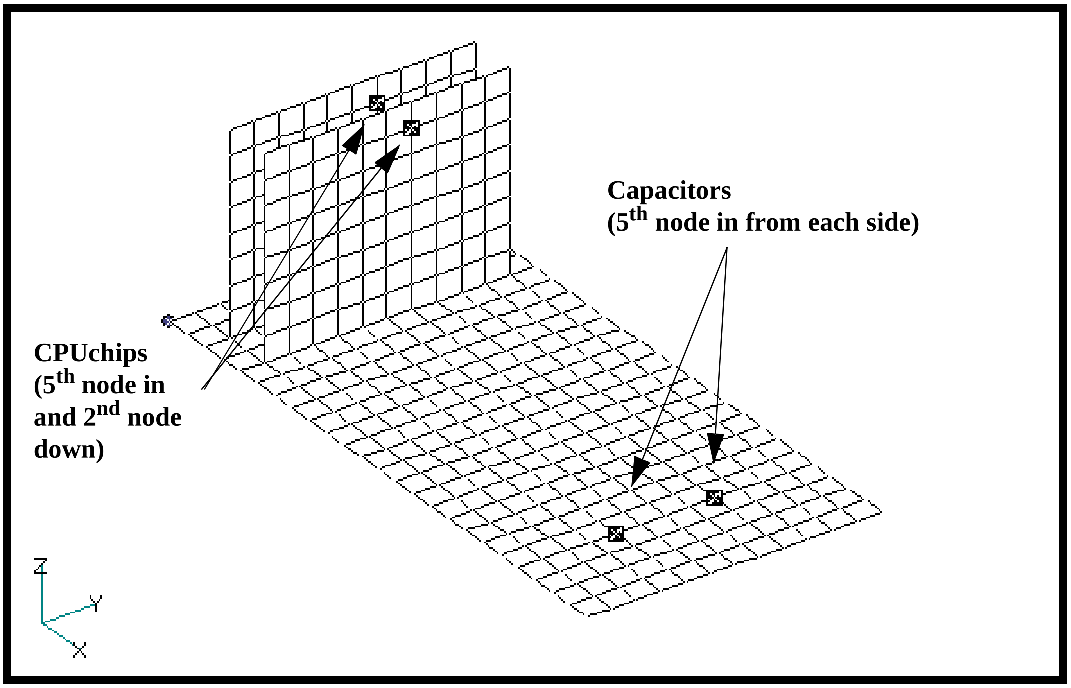
View/Options... (F6)

Quick Options... Ctrl+Q	
Geometry Off	
Done	
OK	

View/Select... (F5)

Model Style:	● **Full Hidden Line**
OK	

Figure 12a.3 - Finite Element of Model



9. Create point masses that will represent capacitors and CPUs. Their locations are shown in Figure 12a.3

Model/Element...

Type...

Other Elements: ● **Mass**

OK

Property: **2.. capacitor**

Node: **161** (see figure 12a.3)

OK

Node: **253** (see figure 12a.3)

OK

Property: **3.. cpu**

Node: **421** (see figure 12a.3)

OK

Node: **520** (see figure 12a.3)

OK

Cancel

10. Equivalence the finite element mesh.

Tools/Check/Coicident Nodes...

Select All

OK

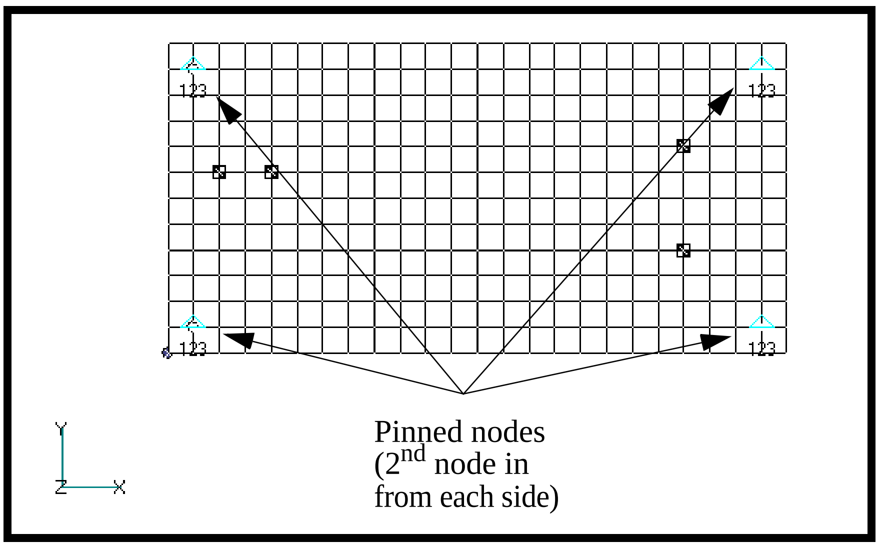
No

Options: ☒ **Merge Coincident Entities**

OK

11. Define the constraints on the model.

Figure 12a.4 - Top Face of Model with Pinned Locations.



Model/Constraint/Set...

Title: **pinned**

OK

View/Rotate... (F8)

XY Top

OK

Model/Constraint/Nodal...

<select nodes 73, 95, 303, 325- see Fig. 12a.4>

OK

Pinned

OK

Cancel

12. Create a new node to store the relative motion calculation and add a constraint equation to calculate the relative motion between nodes in the X direction. This additional node will be added at the top and between the two datughter cards. .

View/Rotate... (F8)

Dimetric

OK

Model/Node...

ID: **1000**

Parameter...

☐ TX ☒ TY ☒ TZ ☒ RX ☒ RY ☒ RZ

OK

Method^ **Between**

On the **Main Tool Bar** select snap to node, ☒

Select the nodes directly above the cpu masses.

Point 1 **348**

Point 2 **447**

OK

Cancel

Now, add the constaint equation. The MPC equation for relative motion is:

$$+1 \times (1000, TX) = +1 \times (447, TX) - 1 \times (348, TX)$$

MSC.Nastran requires the equation to be in the form;

$$0 = +1 \times (1000, TX) + 1 \times (447, TX) - 1 \times (348, TX)$$

Model/Constraint/Equation...

Coefficent	**Node ID**	**DOF**	
1	**1000**	☒ **TX**	**Add**
1	**447**	☒ **TX**	**Add**
-1	**348**	☒ **TX**	**Add**

OK

Cancel

The model is now complete.

13. Submit the file for analysis.

File/Export/Analysis Model...

Type: **2..Normal Modes/Eigenvalue**

OK

Filename: **Circuit_board**

Write

☒ **Run Analysis**

Restarts... ● **Save Databases for Restart**

OK

Advanced...

Eigenvalues and Eigenvectors

Number Desired: **10**

OK

OK

In the **NASTRAN Case Control** window under *Output Requests,* turn all requests off except:

☒ **Displacement**

☒ **Strain Energy**

OK

☒ **WTMASS** **0.00259**

OK

Yes

Save the model.

Filename: **Circuit_board**

Save

Continue

14. Postprocess the results.

View/Select... (F5)

Deformed Style: ● **Deformed**

Contour Style: ● **Contour**

Deformed and Contour Data...

Use the pulldown menus to make the following selections.

Output Set: **1..Mode 1, 19.81138 Hz**

Deformation: **1..Total Translation**

Contour: **80001..Strain Energy Percent**

OK

OK

View/Rotate... (F8)

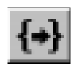
X: **-80**

Y: **0**

Z: **-10**

View each mode in turn. To go through the modes, use the right hand tool bar

then



The frequencies generated are as follows:

Mode	**Frequencies**
1	19.81138
2	37.60053
3	38.75384
4	46.14371
5	47.16561
6	67.60611
7	79.09053
8	108.6912
9	128.4032
10	164.0573

When finished, exit MSC.Nastran for Windows.

File/Exit

Yes

This concludes the exercise