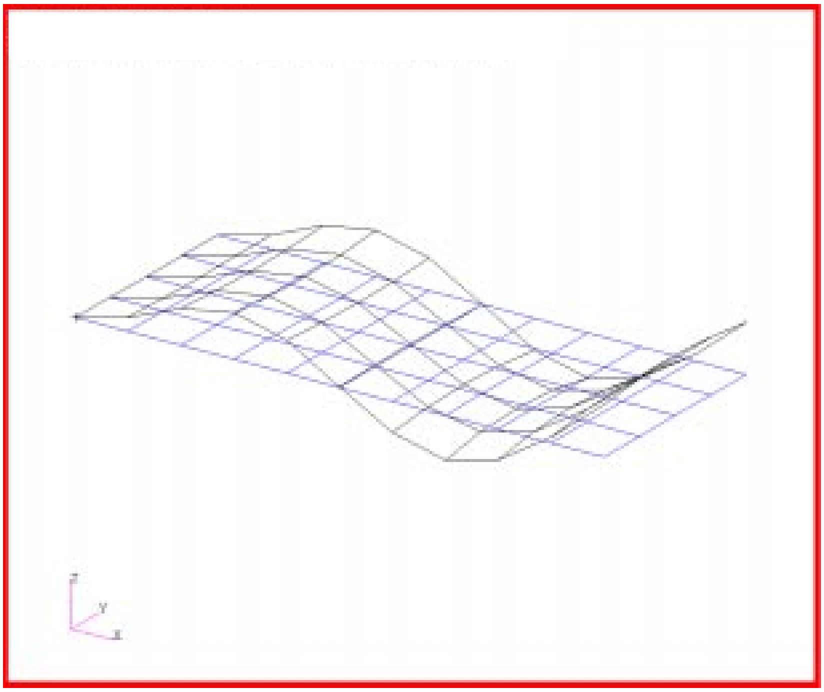
WORKSHOP PROBLEM 1

Modal Analysis of A Flat Plate



Objectives

- Create a geometric representation of a flat rectangular plate.
- Use the geometry model to define an analysis model comprised of plate elements.
- Submit the file for analysis in MSC/NASTRAN.
- Find the first five natural frequencies and mode shapes of the flat plate.

Model Description:

For this example, use Lanczos method to find the first five natural frequencies and mode shapes of a flat rectangular plate. One of the edges is fixed (see Figure 1.2). Below is a finite element representation of the rectangular plate. It also contains the geometric dimensions and the loads and boundary constraints. Table 1.1 contains the necessary parameters to construct the input file.

Figure 1.1 - Grid Coordinates and Element Connectivities

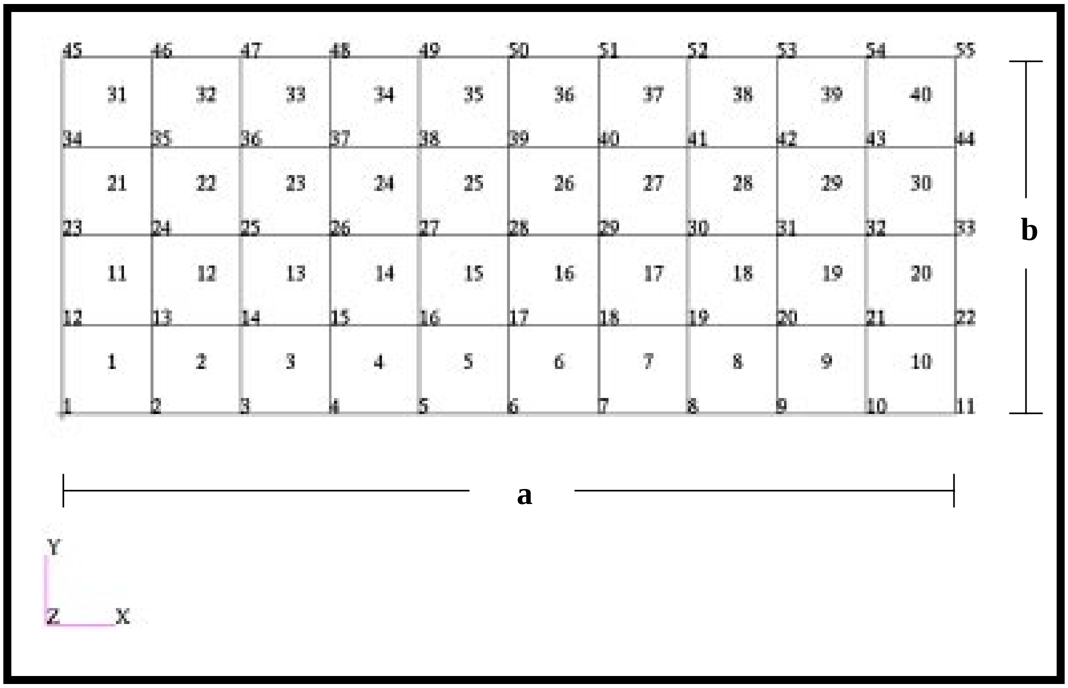


Figure 1.2 - Loads and Boundary Conditions

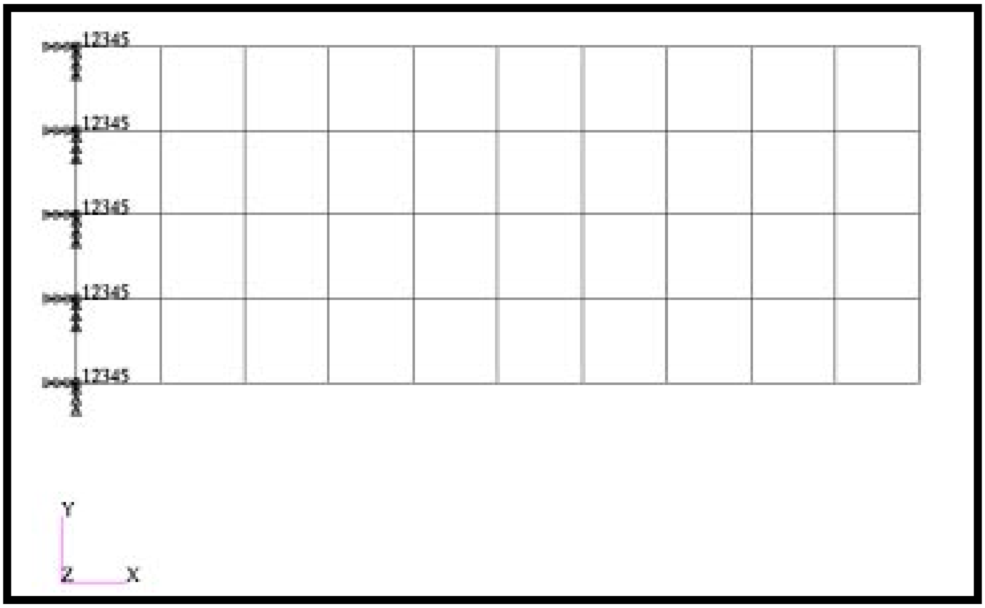


Table 1.1 - Properties

Length (a)	**5 in**
Height (b)	**2 in**
Thickness	**0.100 in**
Weight Density	**0.282 lbs/in^3**
Mass/Weight Factor	**2.59E-3 sec^2/in**
Youngs Modulus	**30.0E6 lbs/in^2**
Poisson's Ratio	**0.3**

Modal Analysis of a Flat Plate

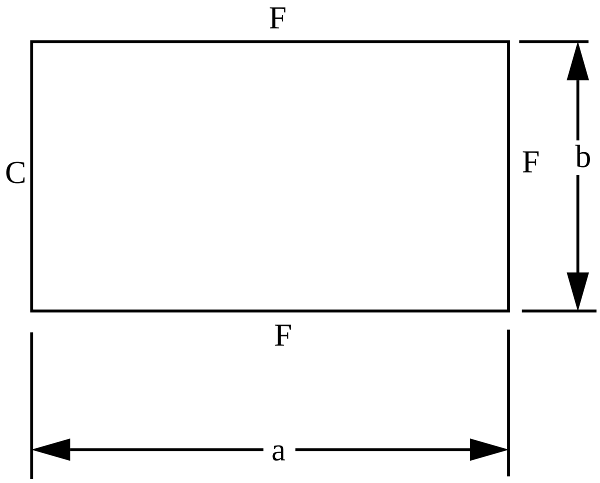
Natural Frequency: Hertz

$$f_{ij} = \frac{\lambda_{ij}^{2}}{2\pi a^{2}}\left[\frac{Eh^{3}}{12\gamma(1-\nu^{2})}\right]^{1/2}$$

where i= 1,2,3, ...

j= 1,2,3, ...

Description: Clamped-Free-Free-Free



a = length of plate

b = width of plate

h = thickness of plate

i = number of half-waves in mode shape along horizontal axis

j = number of half-waves in mode shape along vertical axis

C = clamped edge

E = modulus of elasticity

F = free edge

S = simply supported edge

γ = mass per unit area of plate (μ h for a plate material with density μ)

ν = Poisson ratio

λ_{ij}^2 and (ij)

	Mode Sequence					
a/b	**1**	**2**	**3**	**4**	**5**	**6**
0.40	3.511	4.786	8.115	13.88	21.64	23.73
	(11)	(12)	(13)	(14)	(21)	(22)
2/3	3.502	6.406	14.54	22.04	26.07	31.62
	(11)	(12)	(13)	(21)	(22)	(14)
1.0	3.492	8.525	21.43	27.33	31.11	54.44
	(11)	(12)	(21)	(13)	(22)	(23)
1.5	3.477	11.68	21.62	39.49	53.88	61.99
	(11)	(12)	(21)	(22)	(13)	(31)
2.5	3.456	17.99	21.56	57.46	60.58	106.5
	(11)	(12)	(21)	(22)	(31)	(32)

$\nu = 0.3$

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows 3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

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

Open Model File:	**New Model**

2. Create a material called **mat_1**.

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

Model/Material...

Title:	**mat_1**
Youngs Modulus:	**30e6**
Poisson's Ratio:	**.3**
Mass Density:	**.282**

OK

Cancel

3. Create a property called **plate** to apply to the members of the plate itself.

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

Model/Property...

Title:	**plate**

To select the material, click on the list icon next to the databox and select **mat_1.**

Material:	**1..mat_1**
Thicknesses, Tavg or T1:	**0.1**

OK

Cancel

4. Create the NASTRAN geometry for plate.

Mesh/Between...

To select the property, click on the list icon next to the databox and select **plate**.

Property:	**1..plate**
Mesh Size/ # Nodes/ Dir 1:	**11**
Mesh Size/ # Nodes/ Dir 2:	**5**

OK

	X:	*Y:*	*Z:*
Corner 1:	**0**	**0**	**0**

OK

Repeat this process for the other 3 corners.

X:	*Y:*	*Z:*	
5	**0**	**0**	**OK**
5	**2**	**0**	**OK**
0	**2**	**0**	**OK**

5. To fit the display onto the screen, use the Autoscale feature.

View/Autoscale

6. Create the model constraints.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

Model/Constraint/Set...

Title:	**constraint**

OK

Now define the relevant constraint for the model.

Model/Constraint/Nodal...

Select the five nodes, **Nodes 1, 12, 23, 34, and 45** along the left edge. (Hint: Use shift and left mouse button for rectangular picking.)

OK

On the *DOF* box, select these translational and rotational D.O.F.

☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY**

OK

Cancel

View/Rotate...

Dimetric

OK

7. Create the input file for analysis.

File/Export/Analysis Model...

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

OK

Change the directory to **C:\temp**.

File name: **prob1**

Write

☒ **Run Analysis**

Advanced...

Modal Solution Method: ● **Lanczos**

Eigenvalues and Eigenvectors/ Number Desired: **5**

Mass: ● **Coupled**

OK

Problem ID: **Modal Analysis of a Plate**

OK

Under *Output Requests*, unselect all except:

☒ **Displacement**

OK

Under *PARAM*, enter the following:

☒ *WTMASS* **.00259**

OK

8. When asked if you wish to save the model, respond **Yes**.

Yes

File name: **prob1**

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

Continue

9. Plot the desired mode shapes.

To get a better view of the deformation, you will want to rotate the model as follows:

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

Click on the Output Set listbox to get all modal frequencies, and answer the following questions:

What is the frequency for:

Mode 1 = _______
Mode 2 = _______
Mode 3 = _______
Mode 4 = _______
Mode 5 = _______

To view a mode shape, select one of the modes. View the results of the analysis.

OK

OK

When finished, reset the graphic and exit MSC/NASTRAN for Windows.

View/Select...

Deformed Style: ● **None-Model only**

OK

File/Exit

This concludes this exercise.

Mode 1	**133.70 Hz**
Mode 2	**689.79 Hz**
Mode 3	**843.89 Hz**
Mode 4	**2212.03 Hz**
Mode 5	**2379.05 Hz**