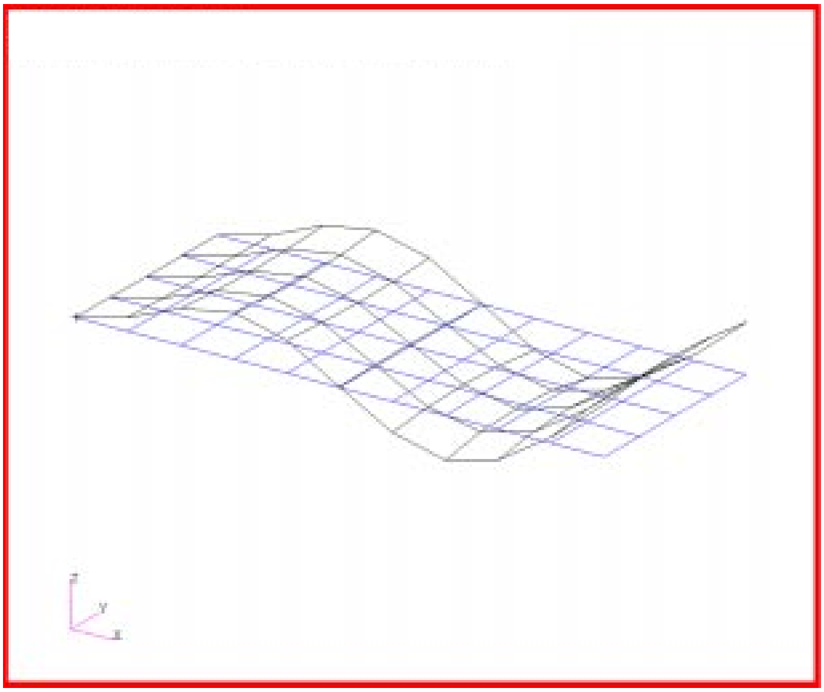
WORKSHOP PROBLEM 1

Modal Analysis of a Flat Plate



Objectives

- Produce a MSC/NASTRAN input file.
- 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

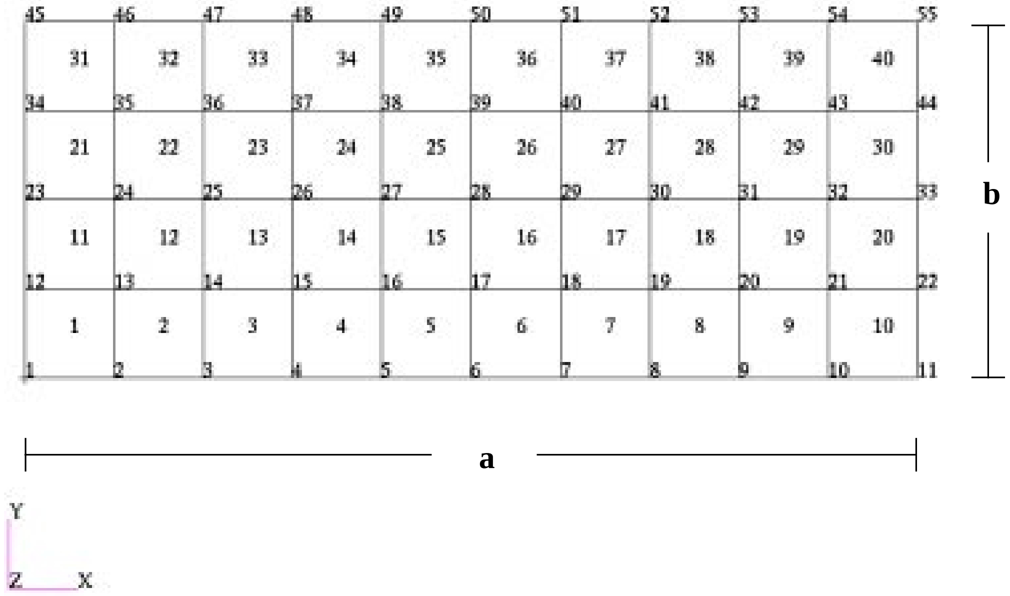


Figure 1.2-Loads and Boundary Conditions

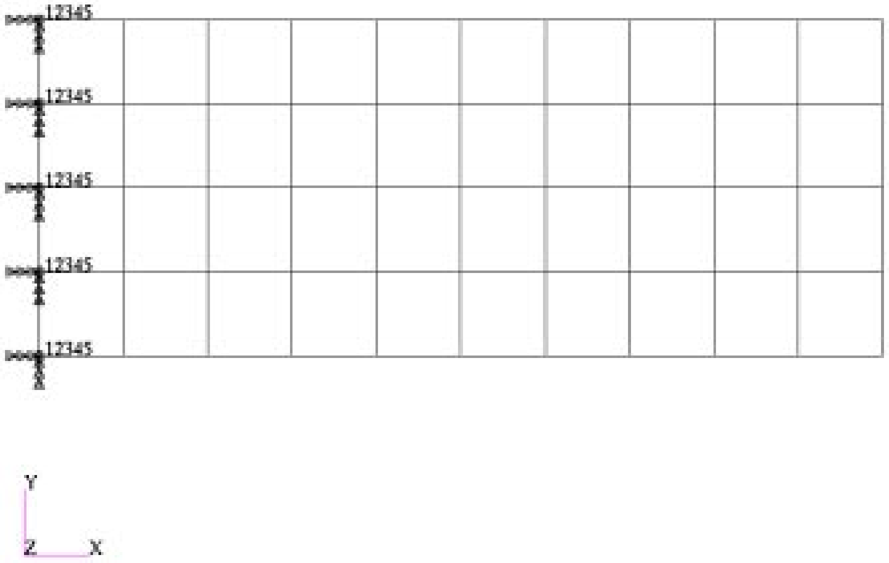


Table 1.1

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**
Elastic Modulus	**30.0E6 lbs/in^2**
Poisson's Ratio	**0.3**

Modal Analysis of a Flat Plate

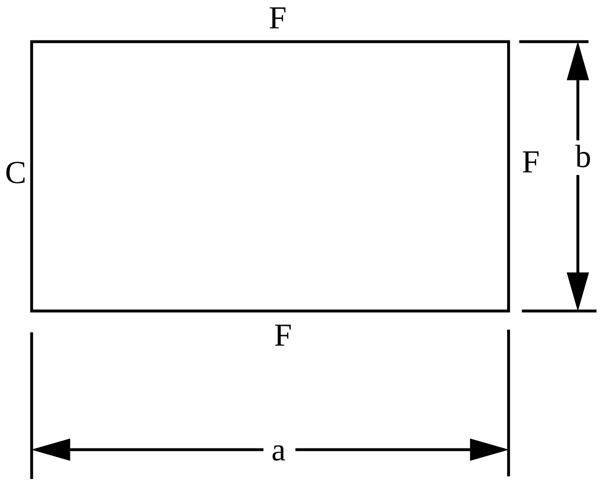
Natural Frequency: Hertz

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

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)

$\nu = 0.3$

Suggested Exercise Steps

- Explicitly generate a finite element representation of the plate structure. (i.e., The grids (GRID) and element connectivities (CQUAD4) should be defined manually.)
- Define material (MAT1) and element (PSHELL) properties.
- Apply the fixed boundary constraints (SPC1).
- Prepare the model for a normal modes analysis (SOL 103 and PARAMs).
  - PARAM, WTMASS, 0.00259
  - PARAM, COUPMASS, 1
- Generate an input file and submit it to the MSC/NASTRAN solver for normal modes analysis.
- Review the results, specifically the eigenvalues.

```
ID SEMINAR,PROB1
```

```
CEND
```

```
BEGIN BULK
```

1	2	3	4	5	6	7	8	9	10

1	2	3	4	5	6	7	8	9	10

ENDDATA

Exercise Procedure:

1. Users who are not utilizing MSC/PATRAN for generating an input file should go to Step 11, otherwise, proceed to step 2.

2. Create a new database named **prob1.db**.

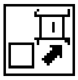
   File/New Database

New Database Name	**prob1**
OK	

   In the *New Model Preference* form set the following:

Tolerance	◆ **Default**
Analysis Code:	**MSC/NASTRAN**
OK	

3. Activate the entity labels by selecting the Show Labels icon on the tool-bar.

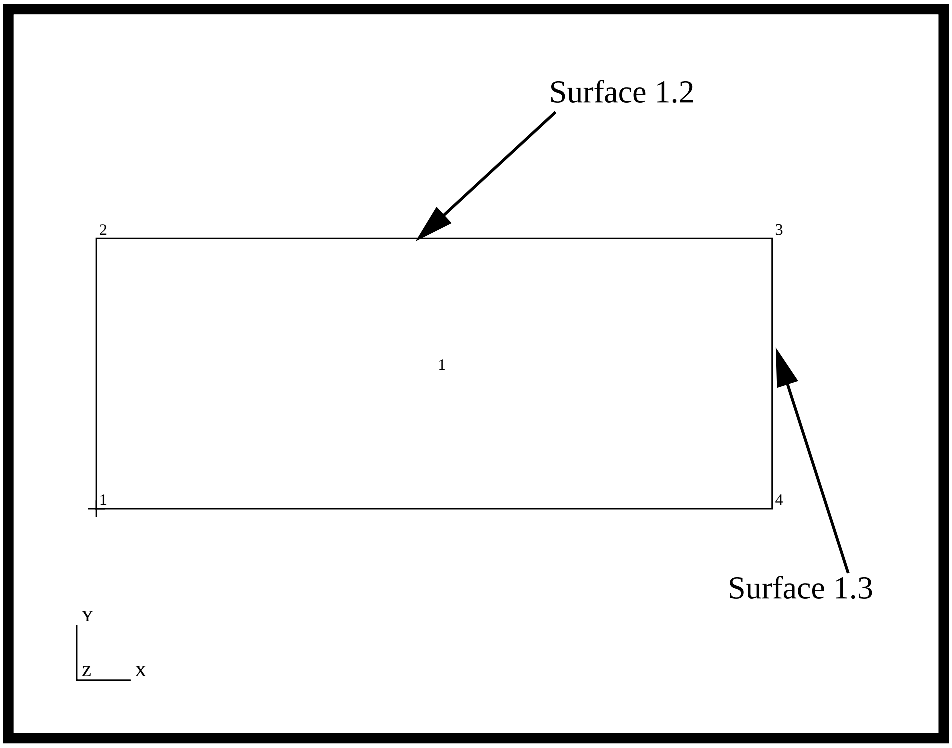
   
   Show Labels

4. Create a surface.

   ◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method	**XYZ**
Vector Coordinates List	**<5, 2, 0>**
Origin Coordinates List	**[0, 0, 0]**
Apply	

Figure 1.3-The surface should resemble the output below.



5. Create the finite element model and mesh the surface.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**

◆ **Number of Elements**

Number =	**10**
Curve List *(see Figure 1.3)*	**Surface 1.2**

Apply

5a. Change the number of mesh seeds to 4 and select the right edge.

Number =	**4**
Curve List *(see Figure 1.3)*	**Surface 1.3**

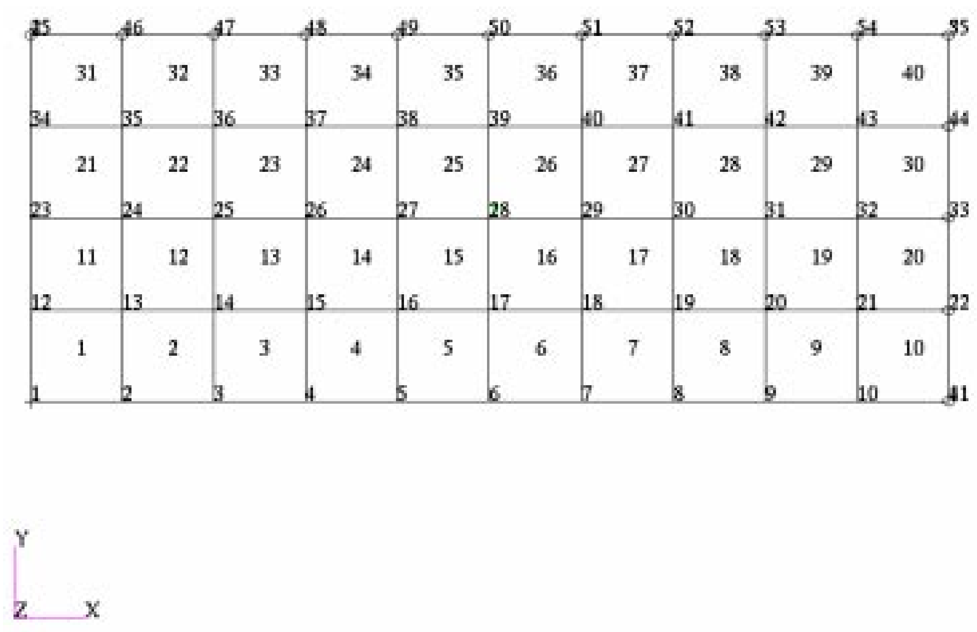
Apply

5b. Mesh the surface.

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Surface List	**Surface 1**

Apply

Figure 1.4-The model should appear as below.



6. Create a set of material properties for the plate.

◆ **Materials**

Action:	**Create**

Object:	**Isotropic**
Method:	**Manual Input**
Material Name	**mat_1**

Input Properties...

Elastic Modulus =	**30.0E6**
Poisson Ratio =	**.3**
Density =	**.282**

Apply

Cancel

7. Define the plate thickness.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**plate**

Input Properties...

Material Name *(Select from **Material Property Sets** box.)*	**m:mat_1**
Thickness	**0.100**

OK

Select Members	**Surface 1**

Add

Apply

8. Apply constraints to the model.

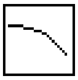
8a. Constrain the left edge from moving through all degrees of freedom.

◆ **Load/BC's**

Action:	**Create**

Object:	**Displacement**
Type:	**Nodal**
New Set Name	**fixed**
Input Data...	
Translations <T1 T2 T3>	**<0, 0, 0>**
Rotations <R1 R2 R3>	**<0, 0, >**
Analysis Coordinate Frame	**Coord 0**
OK	
Select Application Region...	

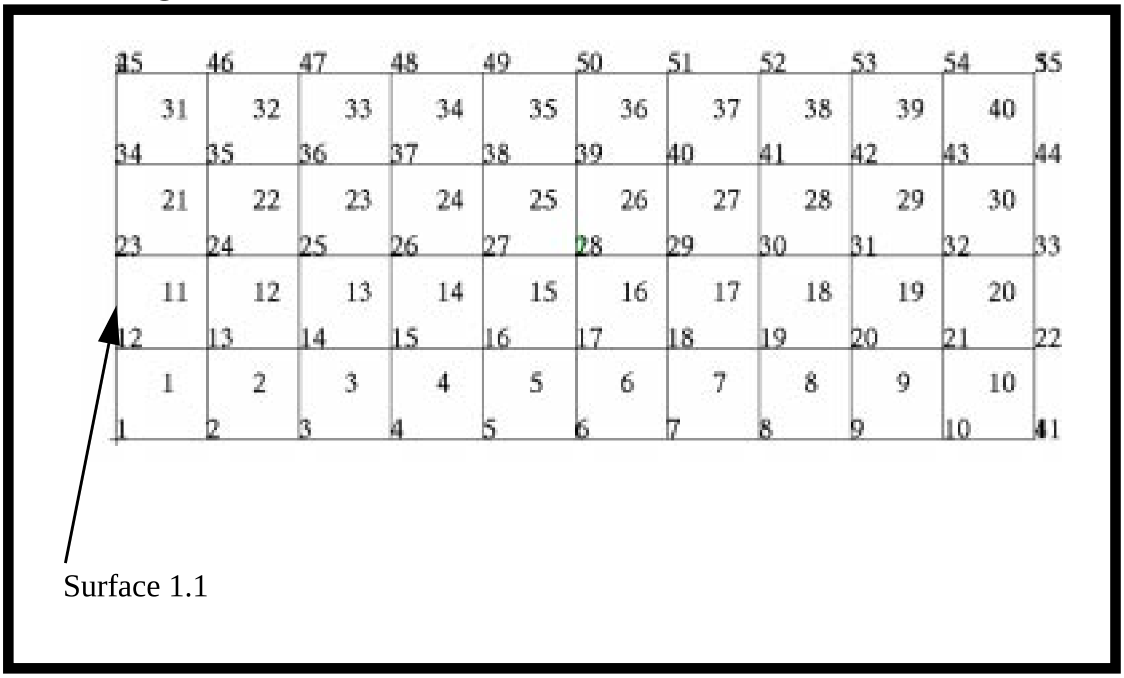
Select the curve or edge icon.



Curve

Select Geometry Entities *(see Figure 1.5)*	**Surface 1.1**
Add	
OK	
Apply	

Figure 1.5



9. Run the analysis.

Before the complete input deck is generated for this analysis, a file that contains only the model data needs to be created. This file is to be used in later workshops.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Model Only**
Job Name	**plate**

Apply

10. Now, you will generate the input file for analysis.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method	**Analysis Deck**

Job Name	**prob1**
Solution Type...	
Solution Type:	**◆ NORMAL MODES**
Solution Parameters...	
Mass Calculation:	**Coupled**
Data Deck Echo:	**Unsorted**
Wt. -Mass Conversion =	**.00259**
OK	
OK	
Subcase Create...	
Available Subcases	**Default**
Subcase Parameters...	
Number of Desired Roots =	**5**
OK	
Output Requests...	

Under ***Output Requests***, highlight:

SPCFORCES(SORT1,Real)=All FEM

Delete

OK

Apply

Cancel

Apply

An MSC/NASTRAN input file called **prob1.bdf** will be generated. The process of translating your model into an input file is called Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC/PATRAN Users should proceed to step 12.

Generating an input file for MSC/NASTRAN Users:

MSC/NASTRAN users can generate an input file using the data from Table 1.1. The result should be similar to the output below.

11. MSC/NASTRAN Input File: **prob1.dat**

```
ID SEMINAR, PROB1
SOL 103
TIME 600
CEND
TITLE = NORMAL MODES EXAMPLE
ECHO = UNSORTED
SUBCASE 1
   SUBTITLE= USING LANCZOS
   METHOD = 1
   SPC = 1
   VECTOR=ALL
BEGIN BULK
PARAM   COUPMASS 1
PARAM    WTMASS  .00259
EIGRL    1                       5
PSHELL   1       1       .1      1               1
CQUAD4   1       1       1       2       13      12
=,*1,=,*1,*1,*1,*1
=8
CQUAD4   11      1       12      13      24      23
=,*1,=,*1,*1,*1,*1
=8
CQUAD4   21      1       23      24      35      34
=,*1,=,*1,*1,*1,*1
=8
CQUAD4   31      1       34      35      46      45
=,*1,=,*1,*1,*1,*1
=8
MAT1     1       3.+7            .3      .282
GRID     1               0.      0.      0.
=,*1,=,*0.5,==
=9
GRID     12              0.      .5      0.
=,*1,=,*0.5,==
=9
GRID     23              0.      1.      0.
=,*1,=,*0.5,==
=9
```

```
GRID     34              0.      1.5     0.
=,*1,=,*0.5,==
=9
GRID     45              0.      2.      0.
=,*1,=,*0.5,==
=9
SPC1     1       12345  1       12      23      34      45
ENDDATA
```

11a. We will also create an input file **plate.bdf**, which contains all the relevant model data. This file is to be used in later workshops.

```
GRID    1               0.      0.      0.
=,*1,=,*0.5,==
=9
GRID    12              0.      .5      0.
=,*1,=,*0.5,==
=9
GRID    23              0.      1.      0.
=,*1,=,*0.5,==
=9
GRID    34              0.      1.5     0.
=,*1,=,*0.5,==
=9
GRID    45              0.      2.      0.
=,*1,=,*0.5,==
=9
PSHELL  1       1       .1      1               1
CQUAD4  1       1       1       2       13      12
=,*1,=,*1,*1,*1,*1
=8
CQUAD4  11      1       12      13      24      23
=,*1,=,*1,*1,*1,*1
=8
CQUAD4  21      1       23      24      35      34
=,*1,=,*1,*1,*1,*1
=8
CQUAD4  31      1       34      35      46      45
=,*1,=,*1,*1,*1,*1
=8
MAT1    1       3.+7            .3      .282
SPC1    1       12345   1       12      23      34      45
```

Submitting the input file for analysis:

12. Submit the input file to MSC/NASTRAN for analysis.

    12a. To submit the MSC/PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob1.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    12b. To submit the MSC/NASTRAN **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob1 scr=yes**. Monitor the run using the UNIX **ps** command.

13. When the run is completed, edit the **prob1.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether existing WARNING messages indicate modeling errors.

14. While still editing **prob1.f06**, search for the word:

R E A L (spaces are necessary)

1st = ___________Hz

2nd = ___________Hz

3rd = ___________Hz

4th = ___________Hz

5th = ___________Hz

Comparison of Results

15. Compare the results obtained in the **.f06** file with the results on the following page:

R E A L   E I G E N V A L U E S

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	7.056994E+05	8.400591E+02	1.336996E+02	1.000000E+00	7.056994E+05
2	2	1.878432E+07	4.334088E+03	6.897916E+02	1.000000E+00	1.878432E+07
3	3	2.811467E+07	5.302327E+03	8.438915E+02	1.000000E+00	2.811467E+07
4	4	1.931709E+08	1.389859E+04	2.212030E+03	1.000000E+00	1.931709E+08
5	5	2.234434E+08	1.494802E+04	2.379052E+03	1.000000E+00	2.234434E+08

16. MSC/NASTRAN Users have finished this exercise. MSC/PATRAN Users should proceed to the next step.

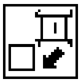
17. Proceed with the Reverse Translation process, that is importing the **prob1.op2** results file into MSC/PATRAN. To do this, return to the Analysis form and proceed as follows:

◆ **Analysis**

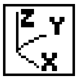
Action:	**Read Output2**
Object:	**Result Entities**
Method	**Translate**
Select Results File...	
Select Results File	**prob1.op2**
OK	

Apply

To simplify the view, turn off the entity labels using the toolbar.

 **Hide Labels**

In addition, switch to a 3 view isometric view point.



Iso 3 View

When the translation is complete bring up the ***Results*** form.

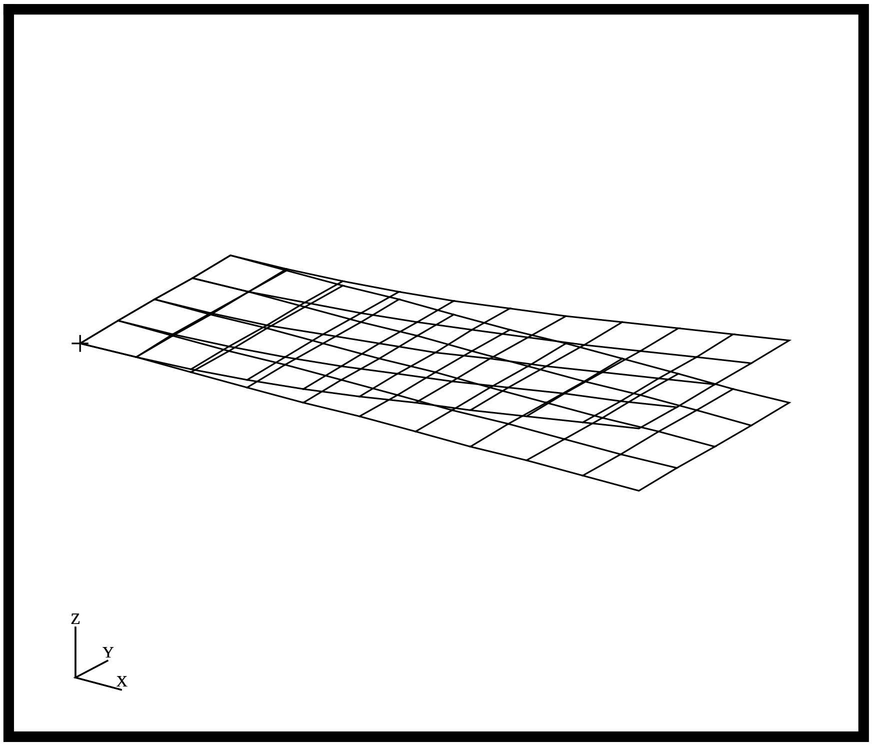
◆ **Results**

Form Type:	**Basic**
Select Results Cases	**1.1-Default, Mode 1**
Select Deformation Result	**1.1 Eigenvectors, Translational**

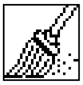
Apply

The results should resemble Figure 1.6.

Figure 1.6



To reset the graphics, click on this icon:

 **Reset Graphics**

Repeat the procedure to view the other mode shapes.

Quit MSC/PATRAN when you are finished with this exercise.