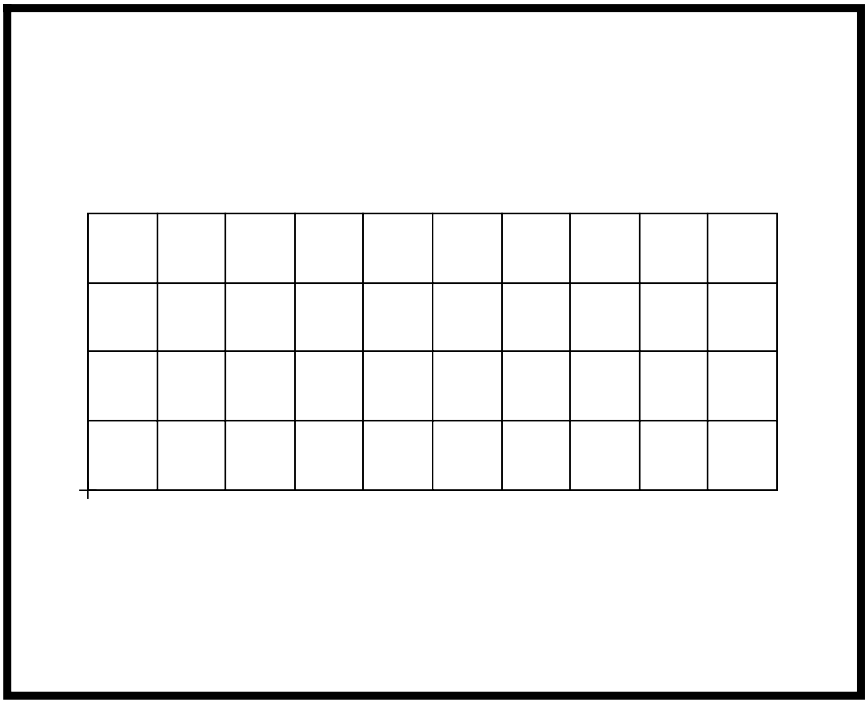
WORKSHOP PROBLEM 7

Elastic Stability of a Plate



Objectives

- Produce a Nastran input file.
- Submit the file for analysis in MSC/NASTRAN.
- Find the first five natural modes of the plate.

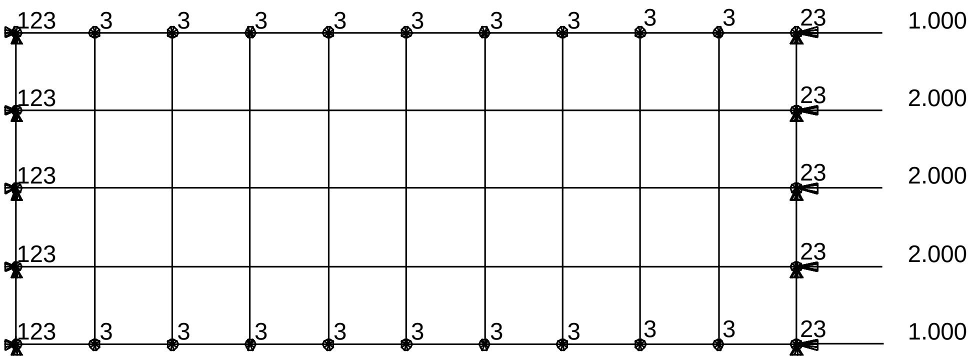
Model Description:

For this example, find the unit critical compressive stress of a flat rectangular plate. This plate is under equal uniform compression of 100 psi on two opposite edges. All edges are simply supported.

In addition, the applied edge compression shall be idealized as nodal forces for this example. See Page 7-4 for helpful hints.

Below is a Finite Element representation of the flat plate. It also contains the geometric dimensions and the loads and boundary constraints. Table 7.1 contains the necessary parameters to construct the input file.

Loads and Boundary Constraints



Grid Coordinates and Element Connectivities

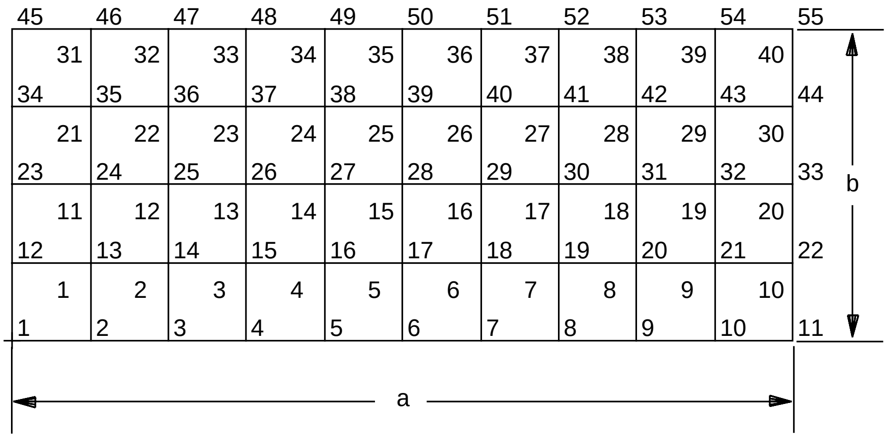


Table 7.1

Elastic Modulus	**29E6 psi**
Poisson Ratio	**0.3**
Plate Thickness	**0.01 in**
Length (a)	**20 in**
Height (b)	**8 in**

Suggested Exercise Steps:

- Explicitly generate a finite element representation of the plate structure i.e., the nodes (GRID) and element connectivity (CQUAD4) should be defined manually.
- Define material (MAT1) and element (PSHELL) properties.
- Apply the simply-supported boundary constraints (SPC1).
- Apply a force load to the model (FORCE).
- Specify real eigenvalue extraction data for Lanczos method (EIGRL).
- Prepare the model for a buckling analysis (SOL 105 and PARAMS).
  - PARAM, COUPMASS, 1
- Generate an Input file and submit it to the MSC/NASTRAN solver for buckling analysis.
- Review the results, specifically the eigenvalues.

hint: conversion of edge pressure to nodal force:

(100 psi)(8 in)(0.01 in) = 8 lbs.

thus:

for 3 middle grids, F = 2 lbs.

for 2 outer grids, F = 1 lb.

```
ID SEMINAR,PROB7
```

```
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 12, otherwise, proceed to step 2.

2. Create a new database called **prob7.db**.

File/New Database

New Database Name:	**prob7**

OK

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

Tolerance:	◆ **Default**
Analysis code:	**MSC/NASTRAN**

OK

3. Next create the geometry for the model.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinate List:	**<20, 8, 0>**

Apply

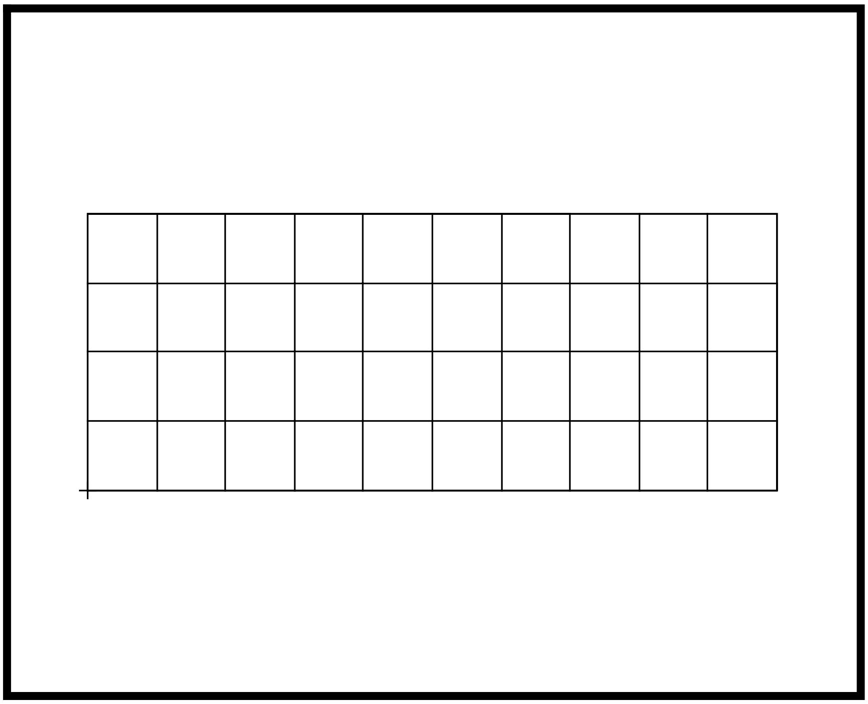
4. Now create the mesh with an edge length of 2.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length:	**2**
Element Topology:	**Quad 4**

Surface List:	**Surface 1**
Apply	

Figure 7.1 - Geometry and Meshing of Plate



5. Create the Material Properties for the plate.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_1**
Input Properties ...	
Elastic Modulus =	**29.0E6**
Poisson Ratio =	**0.3**
Apply	

In the *Current Constitutive Models,* you will see **Linear Elastic - [,,,,] - [Active]** appeared. Click on **Cancel** to close the form.

Cancel

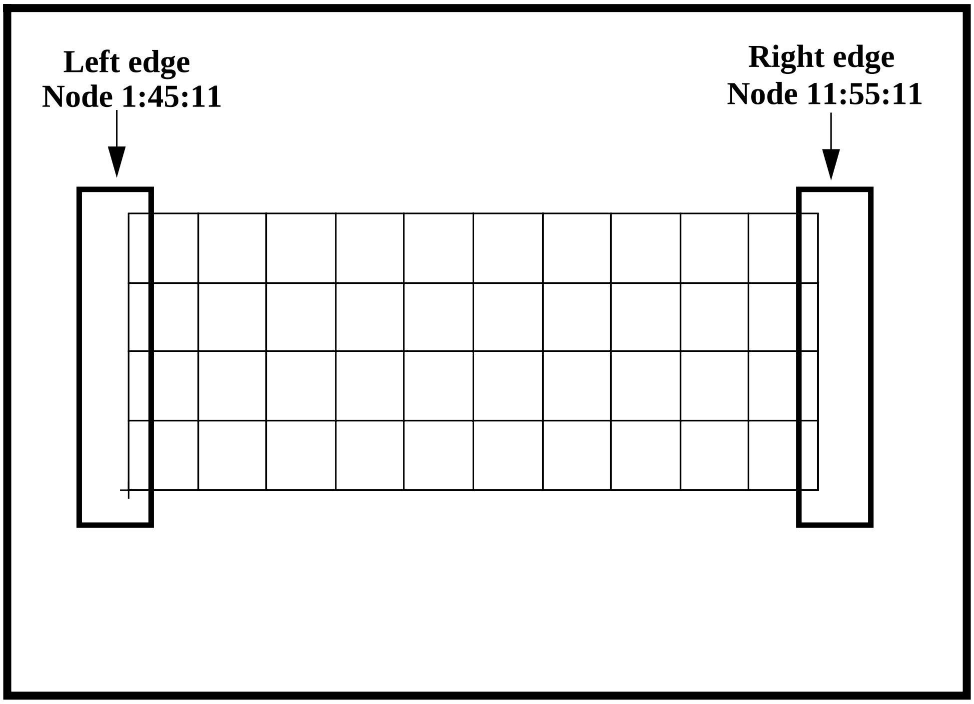
6. Give the plate a thickness using **Properties**.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name:	**plate**
Input Properties ...	
Material Name:	**m:mat_1**
Thickness:	**0.01**
OK	
Select Members:	**Surface 1**
Add	
Apply	

In the next few steps, you will constrain the model.

Figure 7.2 -Node locations



7. First constrain the left edge from moving in the X, Y, Z directions.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**left_edge_constraint**
Input Data...	
Translations <T1 T2 T3>	**<0, 0, 0>**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 1:45:11** (see fig 7.2)
Add	

OK

Apply

8. Next, constrain the right edge from moving in the Y and Z directions.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**right_edge_constraint**

Input Data...

Translations <T1 T2 T3>	**< , 0, 0>**

OK

Select Application Region...

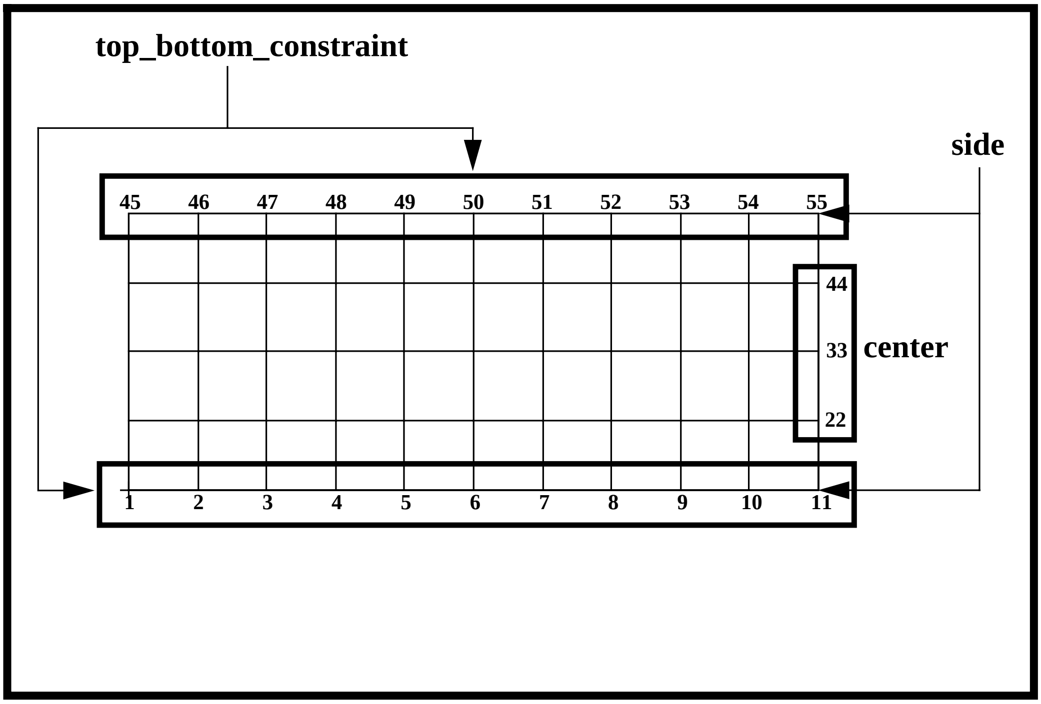
Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 11:55:11** (see fig 7.2)

Add

OK

Apply

Figure 7.3 -Node location



9. Finally, constrain the top and bottom edge from moving in the Z directions.

◆ **Loads/BCs**

Action: **Create**

Object: **Displacement**

Type: **Nodal**

New Set Name: **top_bottom_constraint**

Input Data...

Translations <T1 T2 T3> **< , , 0>**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: **Node 1:11 45:55**
(see fig 7.3)

Add

OK

Apply

10. Now, create the appropriate model loading.

First for the center.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	**center**

Input Data...

Force <F1 F2 F3>	**< -2, 0, 0>**

OK

Select Application Region...

Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 22 33 44** (see fig 7.3)

Add

OK

Apply

Then for the sides.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Force**
Type:	**Nodal**
New Set Name:	**side**

Input Data...

Force <F1 F2 F3>	**< -1, 0, 0>**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: **Node 11 55**

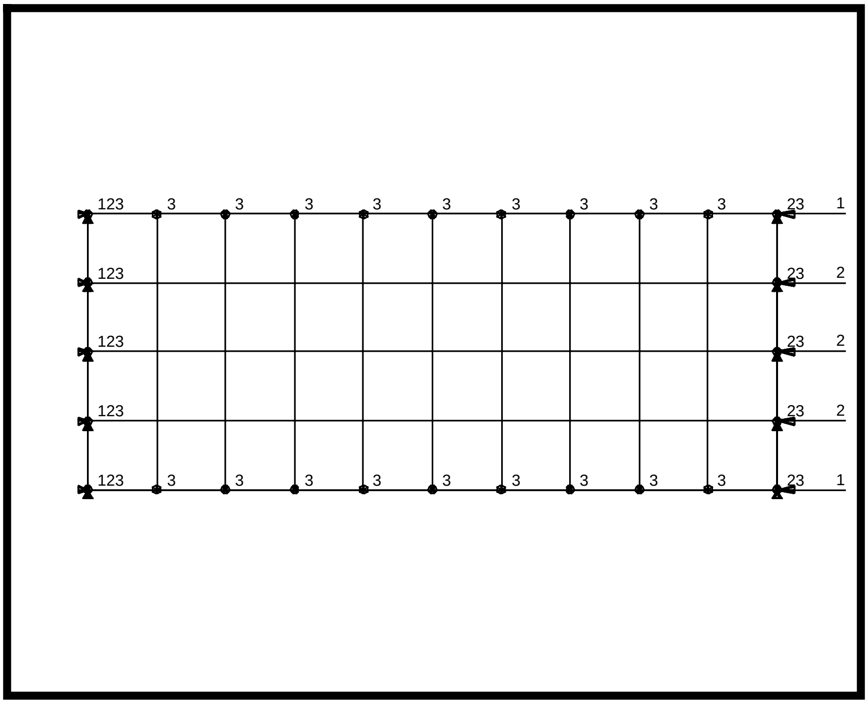
(see fig 7.3)

Add

OK

Apply

Figure 7.4 - Force and Nodal Constraints of Plate



11. Now, we are ready to submit the file for analysis.

◆ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Analysis Deck**

Job Name:	**prob7**
Solution Type...	
Solution Type:	◆ **Buckling**
Solution Parameters...	
Eigenvalue Extraction...	
Number of Desired Roots =	**5**
OK	
OK	
OK	
Apply	

An MSC/NASTRAN input file called **prob7.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC/PATRAN Users should proceed to step 13.

Generating an input file for MSC/NASTRAN Users:

MSC/NASTRAN users can generate an input file using the data from table 7.1. The result should be similar to the output below.

12. MSC/NASTRAN Input File: **prob7.dat**

```
ID SEMINAR, PROB7
SOL 105
TIME 600
CEND
TITLE = ELASTIC STABILITY of PLATES
SUBCASE 1
   SPC = 2
   LOAD = 2
   DISPLACEMENT=ALL
   SPCFORCES=ALL
SUBCASE 2
   SPC = 2
   METHOD = 1
   VECTOR=ALL
   SPCFORCES=ALL
BEGIN BULK
EIGRL    1                       5       0
PSHELL   1       1       .01     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       2.9+7           .3
GRID     1               0.      0.      0.
=        *1      =       *2.     ==
=9
GRID     12              0.      2.      0.
=        *1      =       *2.     ==
=9
GRID     23              0.      4.      0.
=        *1      =       *2.     ==
=9
GRID     34              0.      6.      0.
=        *1      =       *2.     ==
=9
GRID     45              0.      8.      0.
=        *1      =       *2.     ==
=9
```

```
SPCADD  2       1       3       4
LOAD    2       1.      1.      1       1.      3
SPC1    1       123     1       12      23      34      45
SPC1    3       23      11      22      33      44      55
SPC1    4       3       1       THRU    11
SPC1    4       3       45      THRU    55
FORCE   1       11      0       1.      -1.     0.      0.
FORCE   1       55      0       1.      -1.     0.      0.
FORCE   3       22      0       2.      -1.     0.      0.
FORCE   3       33      0       2.      -1.     0.      0.
FORCE   3       44      0       2.      -1.     0.      0.
ENDDATA
```

Submit the input file for analysis

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

    13a.To submit the MSC/PATRAN **.bdf** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob7.bdf scr=yes**. Monitor the run using the UNIX **ps** command.

    13b.To submit the MSC/NASTRAN **.dat** file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran prob7 scr=yes**. Monitor the run using the UNIX **ps** command.

14. When the run is completed, edit the **prob7.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.

15. While still editing **prob7.f06**, search for the word:

E I G E N (spaces are necessary)

Eigenvalue (1st Extraction) = __________________

Comparison of Results:

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

REAL EIGENVALUES

MODE NO.	EXTRACTION ORDER	EIGENVALUE	RADIANS	CYCLES	GENERALIZED MASS	GENERALIZED STIFFNESS
1	1	1.722030E+00	1.312261E+00	2.088529E-01	6.187384E+00	1.065486E+01
2	2	1.759421E+00	1.326432E+00	2.111081E-01	3.294441E+00	5.796309E+00
3	3	2.150348E+00	1.466406E+00	2.333858E-01	1.075235E+01	2.312128E+01
4	4	2.919183E+00	1.708562E+00	2.719260E-01	1.339165E+01	3.909269E+01
5	5	3.733378E+00	1.932195E+00	3.075184E-01	8.730191E-01	3.259310E+00

Since the applied pressure = 8/8(.01) = 100 psi

σ_{cr} = 1.722 (100) = 172.2 psi

Theoretical Value

$$\sigma_{cr} = K\frac{E}{1-v^2}\left(\frac{t}{b}\right)^2$$

Here K depends on the a/b value.

a/b=20/8=2.5,   K=3.373

$$\sigma_{cr} = 3.373\frac{29e6}{1-(0.3)^2}\left(\frac{0.01}{8}\right)^2 = 167.96psi$$

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

18. Proceed with the Reverse Translation process; that is, importing the **prob7.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...

Available Files: **prob7.op2**

OK

Apply

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

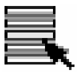
19a. Select **Deformation** to view physical changes of the model.

◆ **Results**

Action: **Create**

Object: **Deformation**

To select results, click on the **Select Results** icon.



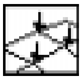
Select Results

Select Results Case: **DEFAULT, Mode 4 : Factor=2.9192**

Select Deformation Result: **Eigenvectors, Translational**

Show As: **Resultant**

To change the Display Attributes, click on the **Display Attributes** icon.

 **Display Attributes**

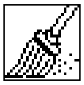
Render Style: **Shaded**

■ **Show Undeformed**

Render Style: **Hidden Line**

Apply

If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the **Reset Graphics** icon.

 **Reset Graphics**

To view different results, after **Reset Graphics** repeat step 23 and change *Result Case(s)* and *Deformation Result.*

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