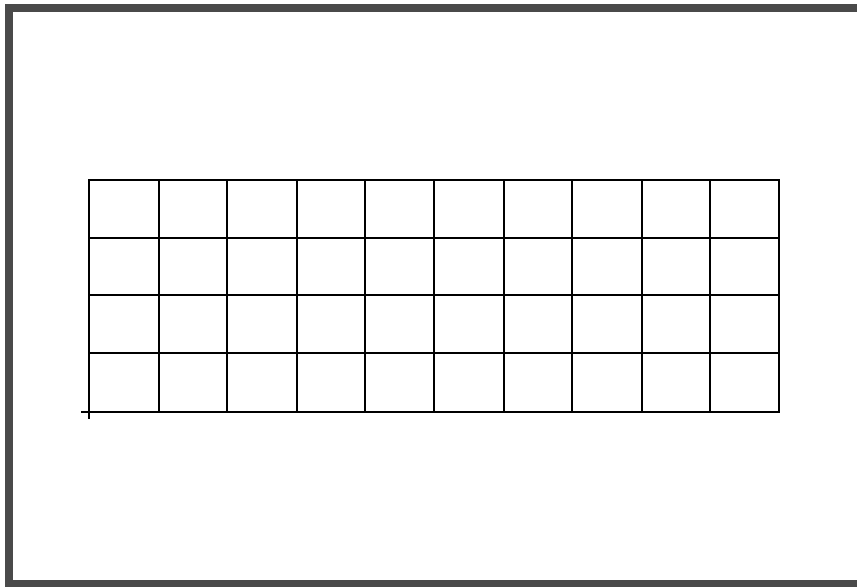


---

## WORKSHOP 5

# *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.



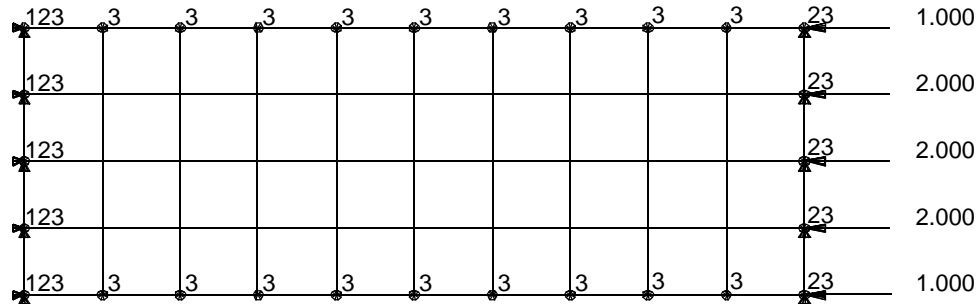
### Model Description:

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.

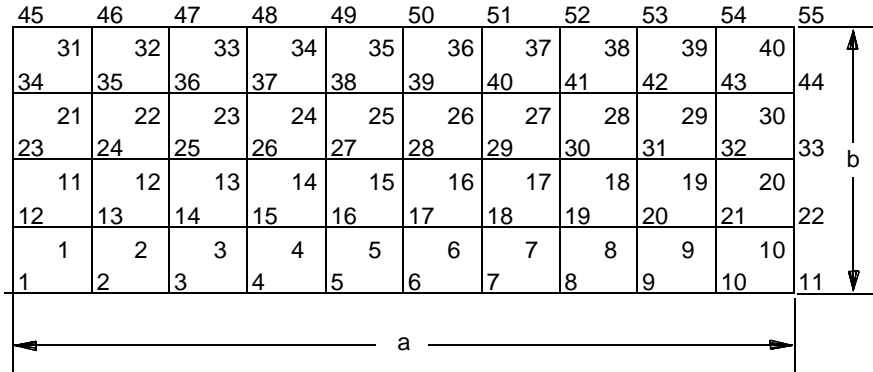
In addition, the applied edge compression shall be idealized as nodal forces for this example. See Page 5-4 for helpful hints.

Below is a Finite Element representation of the flat plate. It also contains the geometric dimensions and the loads and boundary constraints. Table 5.1 contains the necessary parameters to construct the input file.

**Figure 5.1 - Loads and Boundary Constraints**



**Figure 5.2 - Grid Coordinates and Element Connectivities**



**Table 5.1 - Model Properties**

<b>Elastic Modulus:</b>	<b>29E+06 psi</b>
<b>Poisson Ratio:</b>	<b>0.3</b>
<b>Plate Thickness:</b>	<b>0.01 in</b>
<b>Length (a):</b>	<b>20 in</b>
<b>Height (b):</b>	<b>8 in</b>

---

## 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:

$$(100 \text{ psi})(8 \text{ in})(0.01 \text{ in}) = 8 \text{ lbs.}$$

thus,

$$\text{for 3 middle grids, } F = 2 \text{ lbs.}$$

$$\text{for 2 outer grids, } F = 1 \text{ lb.}$$







---

## 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 **prob5.db**.

### File/New...

*New Database Name:*

prob5

OK

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

*Tolerance:*

◆ Default

*Analysis code:*

MSC/NASTRAN

OK

Whenever possible click  **Auto Execute** (turn off).

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

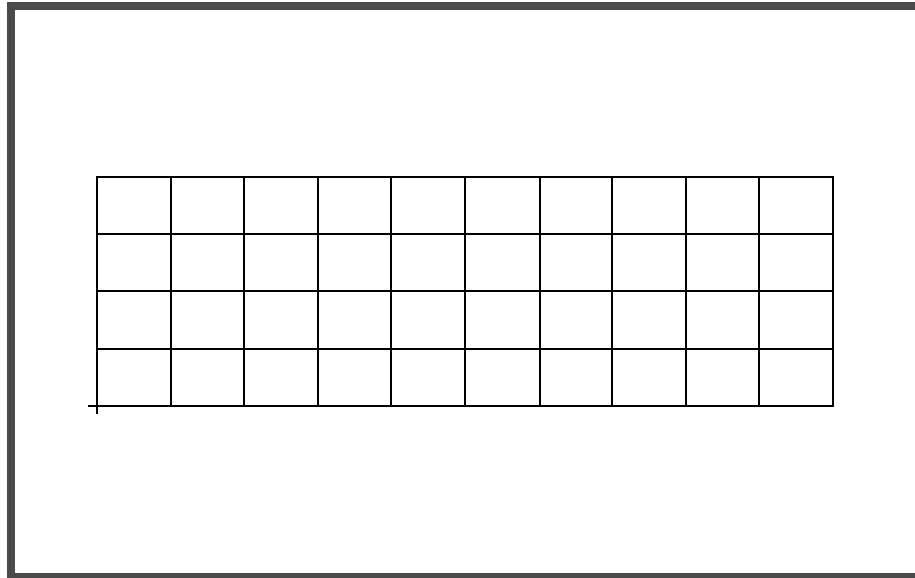
*Global Edge Length:*

2

*Element Topology:*

*Surface List:*

**Figure 5.3** - Geometry and Meshing of Plate



5. Create the Material Properties for the plate.

◆ **Materials**

*Action:*

*Object:*

*Method:*

*Material Name:*

*Elastic Modulus =*

*Poisson Ratio =*

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

---

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:

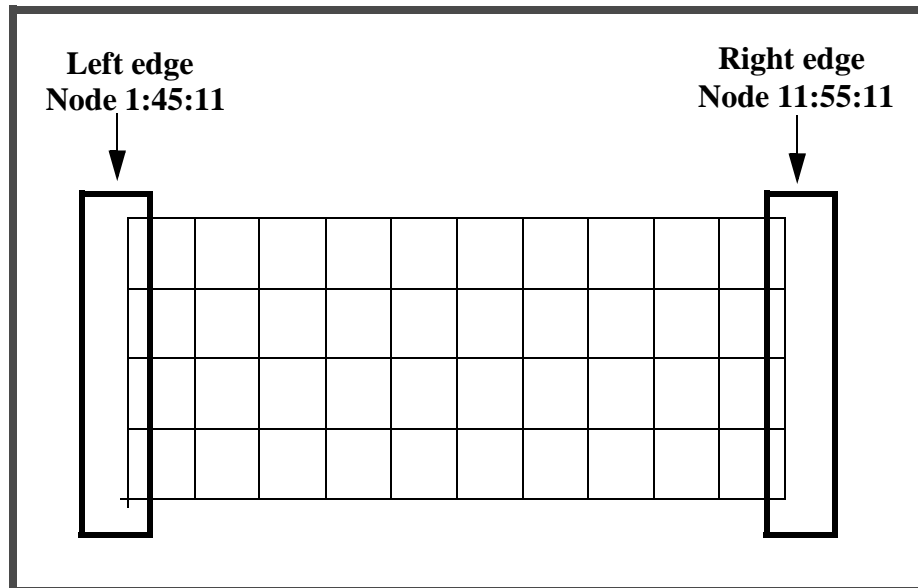
Surface 1

**Add**

**Apply**

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

**Figure 5.4** -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. 5.4)*

**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**

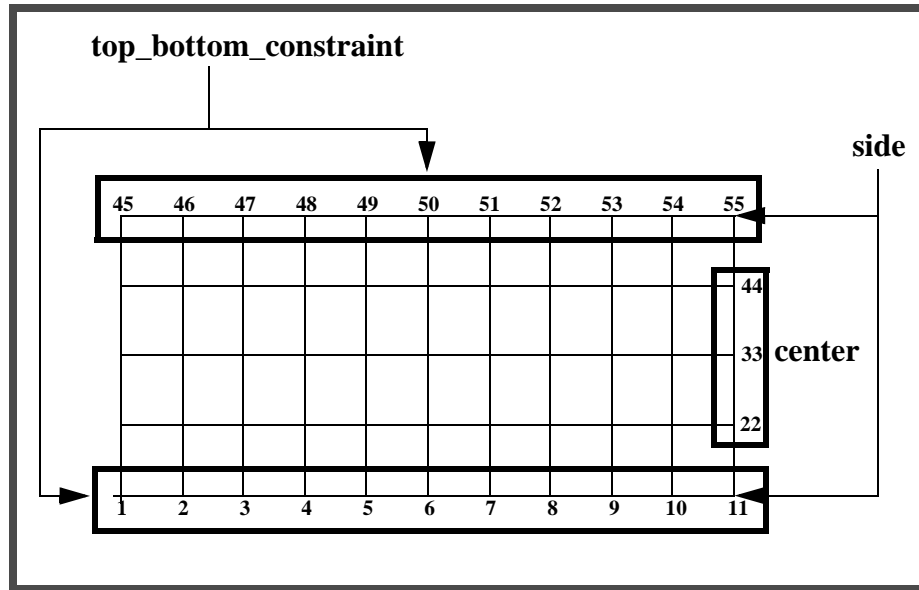
(see Fig. 5.4)

**Add**

**OK**

**Apply**

**Figure 5.5 -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. 5.5)*

**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. 5.5)*

**Add**

**OK**

**Apply**

Then for the sides.

◆ **Loads/BCs**

*Action:*

**Create**

*Object:*

**Force**

Type:

New Set Name:

**Input Data...**

Force <F1 F2 F3>

**OK**

**Select Application Region...**

Geometry Filter:

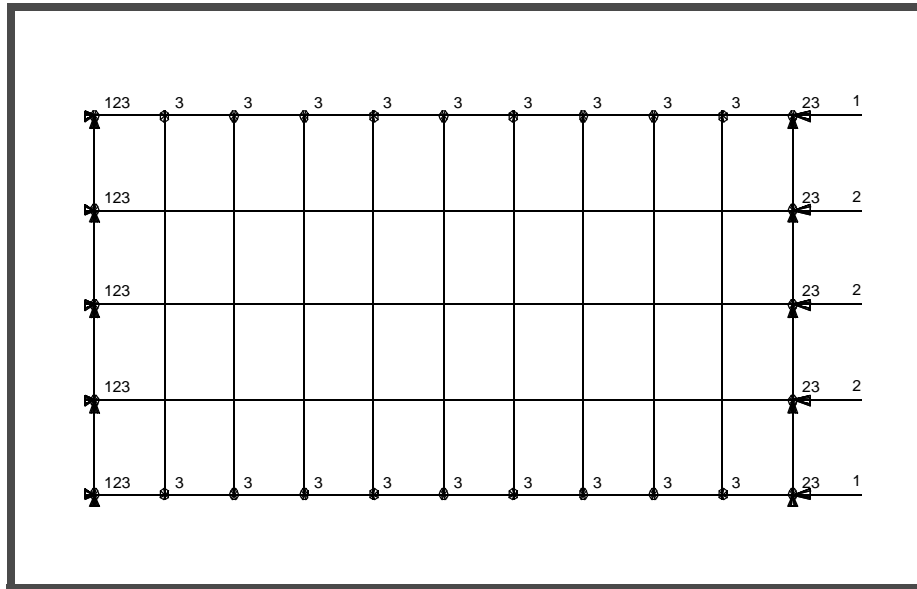
Select Nodes:   
(see Fig. 5.5)

**Add**

**OK**

**Apply**

**Figure 5.6 - Force and Nodal Constraints of Plate**



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

**◆ Analysis**

Action:

Object:

*Method:*

Analysis Deck

*Job Name:*

prob5

Solution Type...

*Solution Type:*

◆ Buckling

Solution Parameters...

Eigenvalue Extraction...

*Number of Desired Roots =*

5

OK

OK

OK

Apply

An MSC.Nastran input file called **prob5.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.

---

# Generating an Input File for MSC.Nastran Users:

MSC.Nastran users can generate an input file using the data from Table 5.1. The result should be similar to the output below.

## 12. MSC.Nastran Input File: **prob5.dat**

```
ID SEMINAR, PROB5
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
    
```

---

## Submitting 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 prob5.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 prob5.dat scr=yes**. Monitor the run using the UNIX **ps** command.
14. When the run is completed, edit the **prob5.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 **prob5.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:

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	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**  
 **$\sigma_{cr} = 1.722 (100) = 172.2$  psi**

Theoretical Value:

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

Here K depends on the a/b value.

$$a/b = 20/8 = 2.5, \quad K = 3.373$$

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

---

**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 **prob5.op2** results file into MSC.Patran. To do this, return to the *Analysis* form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	<b>Read Output2</b>
<i>Object:</i>	<b>Result Entities</b>
<i>Method:</i>	<b>Translate</b>
<b>Select Results File...</b>	
<i>Available Files:</i>	<b>prob5.op2</b>
<b>OK</b>	
<b>Apply</b>	

19. When the translation is complete bring up the **Results** form.  
19a. Select **Deformation** to view physical changes of the model.

◆ **Results**

<i>Action:</i>	<b>Create</b>
<i>Object:</i>	<b>Deformation</b>

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



**Select Results**

<i>Select Results Case:</i>	<b>DEFAULT, Mode 4: Factor=2.9192</b>
<i>Select Deformation Result:</i>	<b>Eigenvectors, Translational</b>
<i>Show As:</i>	<b>Resultant</b>

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 19 and change *Result Case(s)* and *Deformation Result*.

Quit MSC.Patran when you are finished with this exercise.

