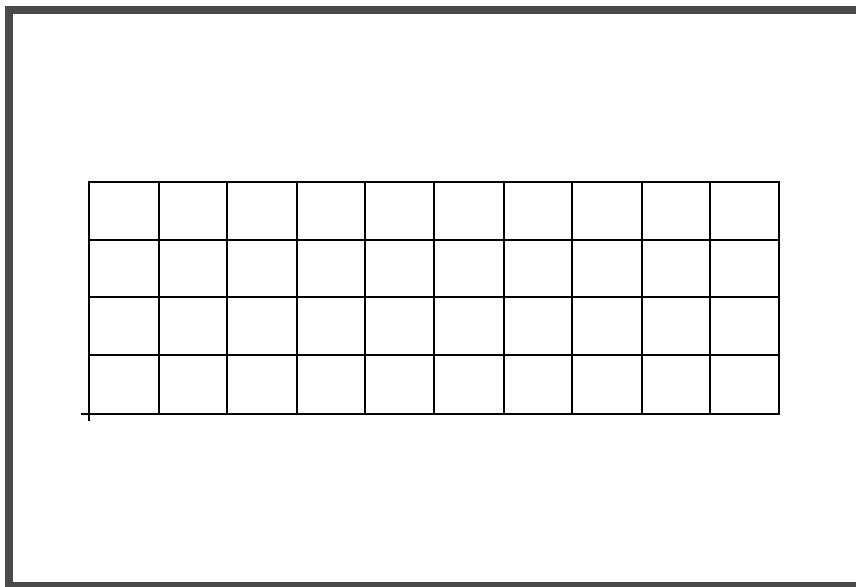

WORKSHOP 7

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 7-5 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.

Figure 7.1 - Loads and Boundary Constraints

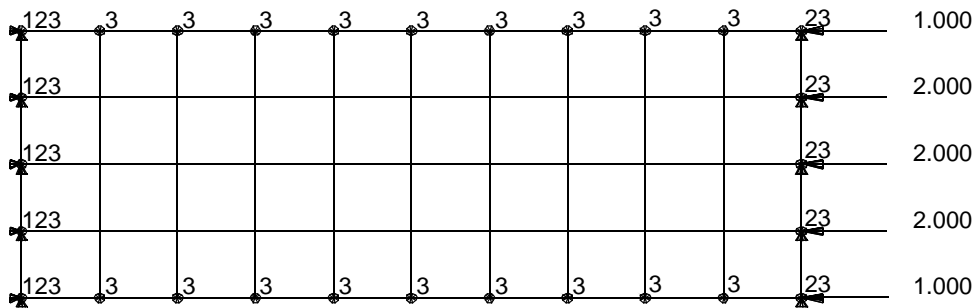


Figure 7.2 - Grid Coordinates and Element Connectivities

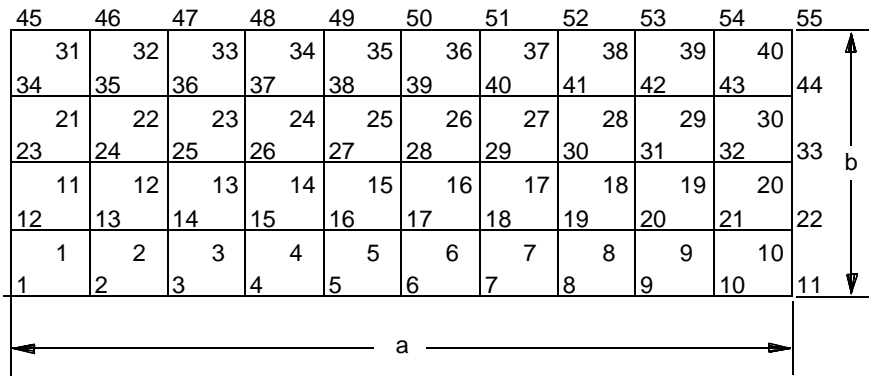
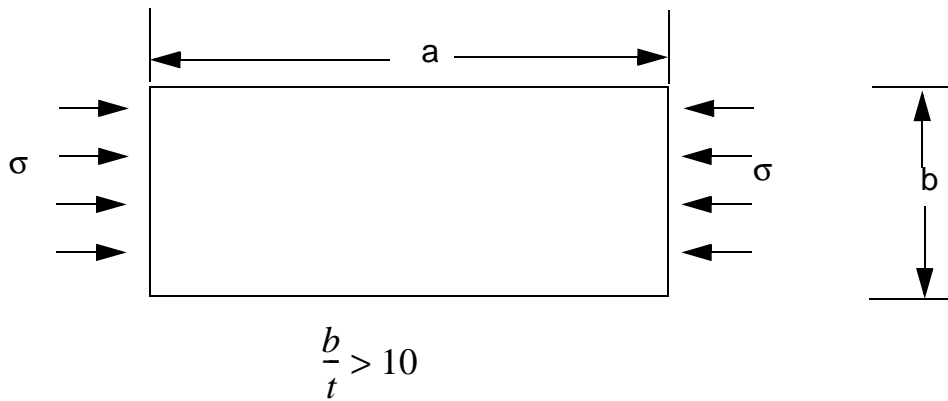


Table 7.1 - Model Properties

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

Formula For elastic stability of plate:



- Rectangular Plate under equal uniform compression on two opposite edges b.
- All edges are simply supported

$$\sigma' = k \frac{E}{1 - \nu^2} \left(\frac{t}{b} \right)^2$$

Since $\frac{a}{b} = \frac{20}{8} = 2.5$

Therefore $k = 3.373$

$$\sigma' = 3.373 \frac{29e6}{1 - (0.3)^2} \left(\frac{0.01}{8} \right)^2 = 168.0 \text{ psi}$$

Reference: Roark and Young. Formula for Stress and Strain, Fifth Edition, McGraw-Hill Book Company

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).
- 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 **prob7.db**.

File/New...

New Database Name:

prob7

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 element edge length of 2.

◆ **Finite Elements**

Action:

Create

Object:

Mesh

Type:

Surface

Global Edge Length:

2

Element Topology:

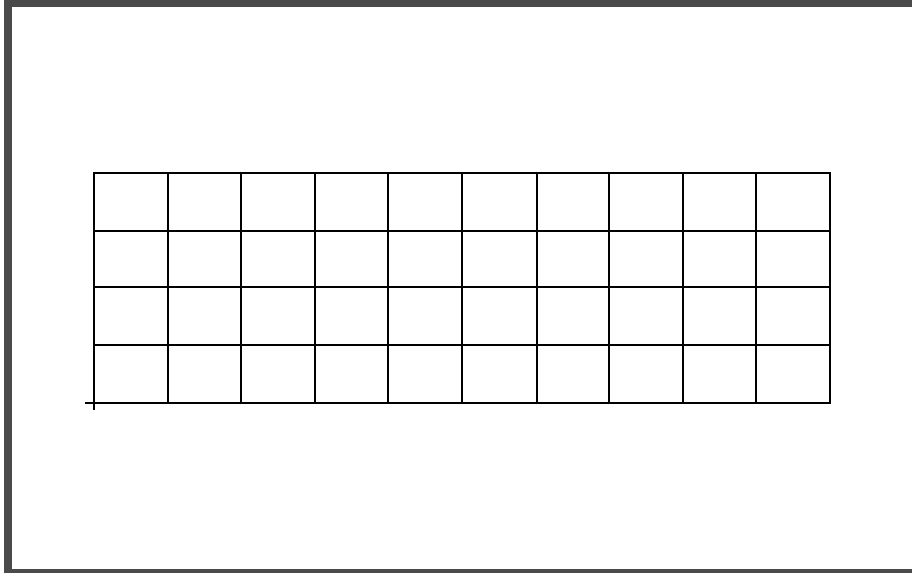
Quad 4

Surface List:

Surface 1

Apply

Figure 7.3 - 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

OK

Apply

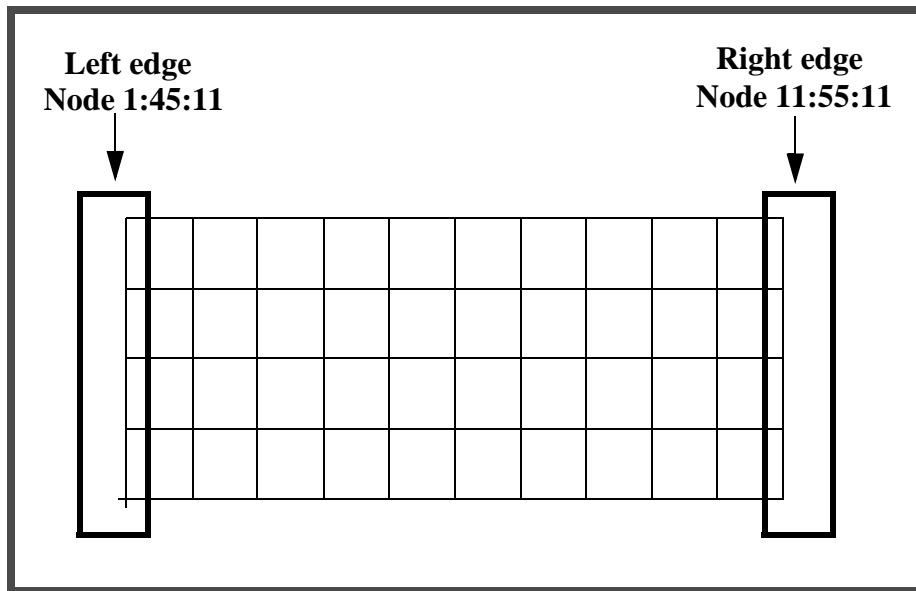
6. Give the plate a thickness using **Properties**.

◆ **Properties**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Dimension:</i>	<input type="text" value="2D"/>
<i>Type:</i>	<input type="text" value="Shell"/>
<i>Property Set Name:</i>	<input type="text" value="plate"/>
<input type="button" value="Input Properties..."/>	
<i>Material Name:</i>	<input type="text" value="m:mat_1"/>
<i>Thickness:</i>	<input type="text" value="0.01"/>
<input type="button" value="OK"/>	
<i>Select Members:</i>	<input type="text" value="Surface 1"/>
<input type="button" value="Add"/>	
<input type="button" value="Apply"/>	

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

Figure 7.4 -Node locations



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

◆ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
----------------	-------------------------------------

Object:
Type:
New Set Name:

Translations <T1 T2 T3>

Geometry Filter:
Select Nodes:
(see Fig. 7.4)

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

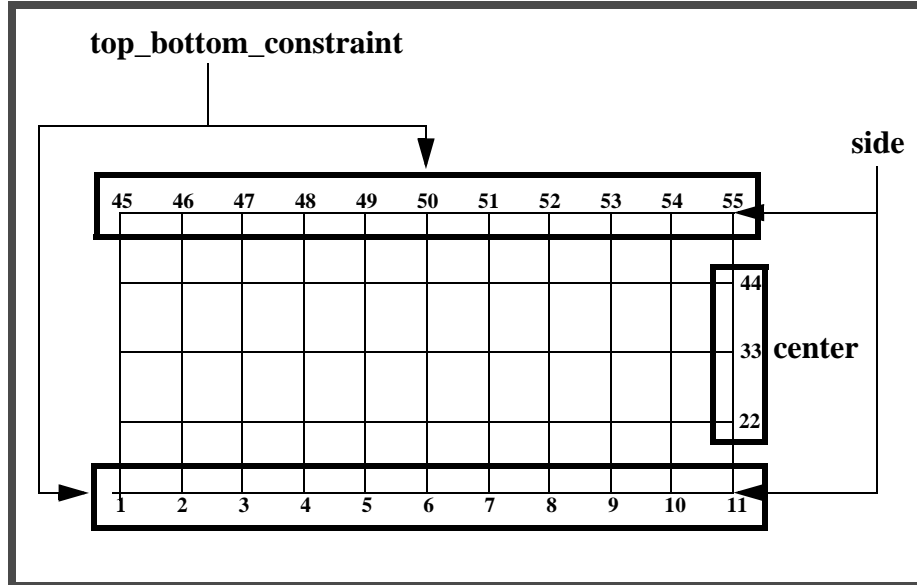
◆ Loads/BCs
Action:
Object:
Type:
New Set Name:

Translations <T1 T2 T3>

Geometry Filter:
Select Nodes:
(see Fig. 7.4)

Apply

Figure 7.5 -Node location



- 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. 7.5)

Add

10. Now, create the appropriate model loading.

First for the center.

◆ **Loads/BCs**

Action:

Object:

Type:

New Set Name:

Force <F1 F2 F3>

Geometry Filter:

◆ **FEM**

Select Nodes:

(see Fig. 7.5)

Then for the sides.

◆ **Loads/BCs**

Action:

Object:

Type:

New Set Name:

Force <F1 F2 F3>

OK

Select Application Region...

Geometry Filter: **◆ FEM**

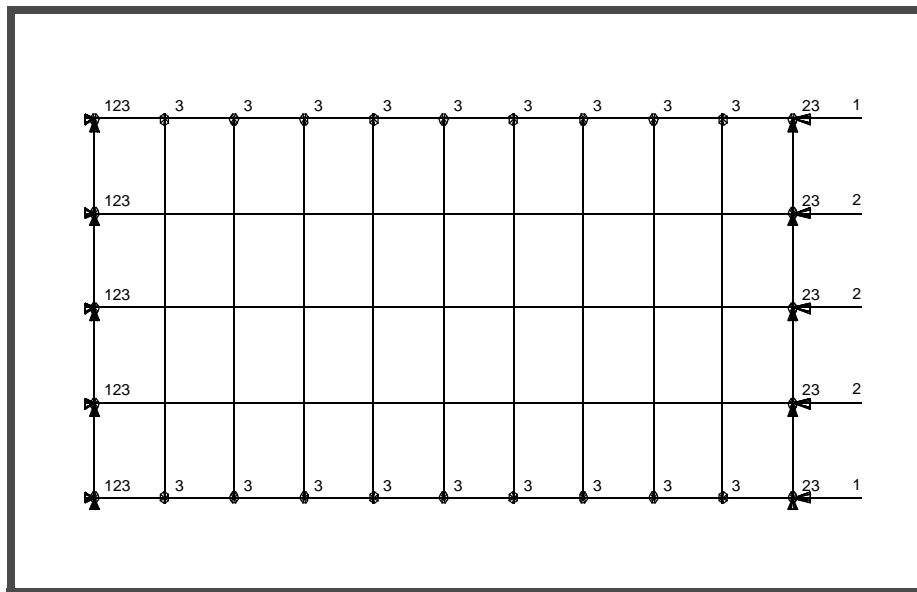
Select Nodes: **Node 11, 55**
(see Fig. 7.5)

Add

OK

Apply

Figure 7.6 - 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**

Translation Parameters...

Data Output: **XDB and Print**

OK

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.

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
PARAM, POST, 0
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 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.dat 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:

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	<u>1.722030E+00</u>	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

**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, attaching the **prob7.xdb** results file into MSC.Patran. To do this, return to the *Analysis* form and proceed as follows:

◆ **Analysis**

Action:

Attach XDB

Object:

Result Entities

Method:

Local

Select Results File...

Available Files:

prob7.xdb

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, A1: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 19 and change *Result Case(s)* and *Deformation Result*.

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

