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.
Elastic Stability of a 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.

<table>
<thead>
<tr>
<th>Load and Boundary Constraints</th>
</tr>
</thead>
<tbody>
<tr>
<td>Grid Coordinates and Element Connectivities</td>
</tr>
</tbody>
</table>

Table 7.1

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Elastic Modulus</td>
<td>29E6 psi</td>
</tr>
<tr>
<td>Poisson Ratio</td>
<td>0.3</td>
</tr>
<tr>
<td>Plate Thickness</td>
<td>0.01 in</td>
</tr>
<tr>
<td>Length (a)</td>
<td>20 in</td>
</tr>
<tr>
<td>Height (b)</td>
<td>8 in</td>
</tr>
</tbody>
</table>
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:**

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

for 2 outer grids, \( F = 1 \text{ lb.}\)
### Elastic Stability of a Plate

<p>| | | | | | | | | | |</p>
<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

**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>

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
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

**Figure 7.1 - Geometry and Meshing of Plate**
In the *Current Constitutive Models*, you will see **Linear Elastic** - [,,] - [**Active**] appeared. Click on **Cancel** to close the form.

<table>
<thead>
<tr>
<th>Cancel</th>
</tr>
</thead>
</table>

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

◆ **Properties**

<table>
<thead>
<tr>
<th>Action:</th>
<th>Create</th>
</tr>
</thead>
<tbody>
<tr>
<td>Dimension:</td>
<td>2D</td>
</tr>
<tr>
<td>Type:</td>
<td>Shell</td>
</tr>
<tr>
<td>Property Set Name:</td>
<td>plate</td>
</tr>
</tbody>
</table>

**Input Properties ...**

| Material Name: | m:mat_1 |
| Thickness: | 0.01 |

<table>
<thead>
<tr>
<th>OK</th>
</tr>
</thead>
</table>

**Select Members:**

| Add |
| Apply |

In the next few steps, you will constrain the model.
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
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
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**
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>
11. Now, we are ready to submit the file for analysis.

◆ **Analysis**

*Action:* Analyze
*Object:* Entire Model
*Method:* Analysis Deck
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:
### Elastic Stability of a Plate

#### Real Eigenvalues

<table>
<thead>
<tr>
<th>MODE NO.</th>
<th>ORDER</th>
<th>EIGENVALUE</th>
<th>RADIIANS</th>
<th>CYCLES</th>
<th>GENERALIZED MASS</th>
<th>GENERALIZED STIFFNESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>1</td>
<td>1.722030E+00</td>
<td>1.312261E+00</td>
<td>2.088529E-01</td>
<td>6.107384E+00</td>
<td>1.065486E+01</td>
</tr>
<tr>
<td>2</td>
<td>2</td>
<td>1.759421E+00</td>
<td>1.326432E+00</td>
<td>2.11081E-01</td>
<td>3.294441E+00</td>
<td>5.796309E+00</td>
</tr>
<tr>
<td>3</td>
<td>3</td>
<td>2.150348E+00</td>
<td>1.466406E+00</td>
<td>2.33858E-01</td>
<td>1.075235E+01</td>
<td>2.312128E+01</td>
</tr>
<tr>
<td>4</td>
<td>4</td>
<td>2.919183E+00</td>
<td>1.708562E+00</td>
<td>2.719260E-01</td>
<td>1.339165E+01</td>
<td>3.909269E+01</td>
</tr>
<tr>
<td>5</td>
<td>5</td>
<td>3.733378E+00</td>
<td>1.932195E+00</td>
<td>3.075184E-01</td>
<td>8.730191E-01</td>
<td>3.259310E+00</td>
</tr>
</tbody>
</table>

Since the applied pressure = \(\frac{8}{8}(0.01) = 100\) psi

\[\sigma_{cr} = 1.722 \times 100 = 172.2\] psi
Theoretical Value

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

Here K depends on the a/b value.

\[ \frac{a}{b} = \frac{20}{8} = 2.5, \quad K = 3.373 \]

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