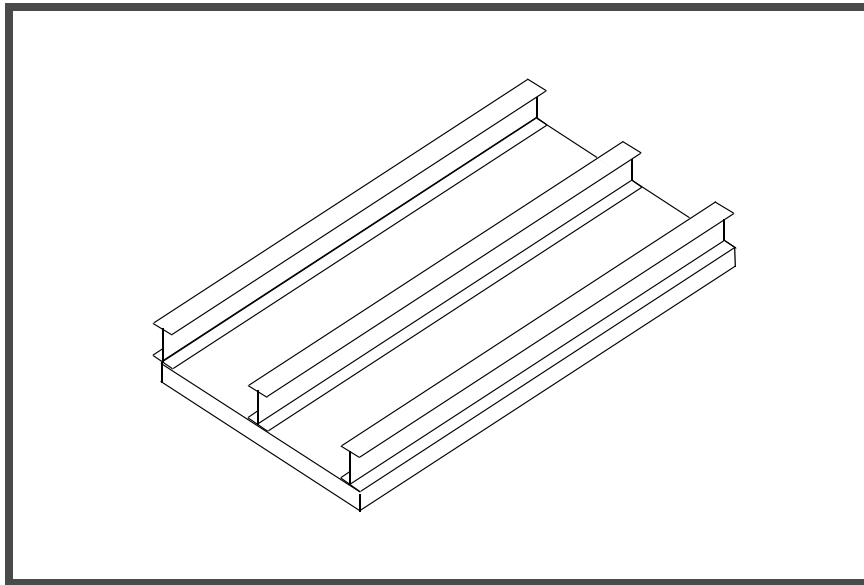

WORKSHOP 3

Linear Static Analysis of a Simply-Supported Stiffened Plate



Objectives:

- Create a MSC.Nastran analysis model comprised of CQUAD4 & CBAR elements.
- Prepare a MSC.Nastran input file for a Linear Static analysis.
- Visualize analysis results.

Model Description:

Below is a finite element representation of the stiffened plate shown on page 3-1. The plate is 0.1 inches thick; therefore thin-shell theory applies. I-beam stiffeners are mounted as shown. The structure has pin supports on its four corners and a uniform pressure of 0.5 psi is applied to the surface of the plate (See Figure 3.3). Table 3.1 contains all the necessary values to define the material and property of the stiffened plate.

HINT: Because the centroidal axes of the stiffeners do not coincide with the mid-plane of the plate, you will need to account for this when you define the element properties for the stiffeners.

Figure 3.1 - Model Schematics

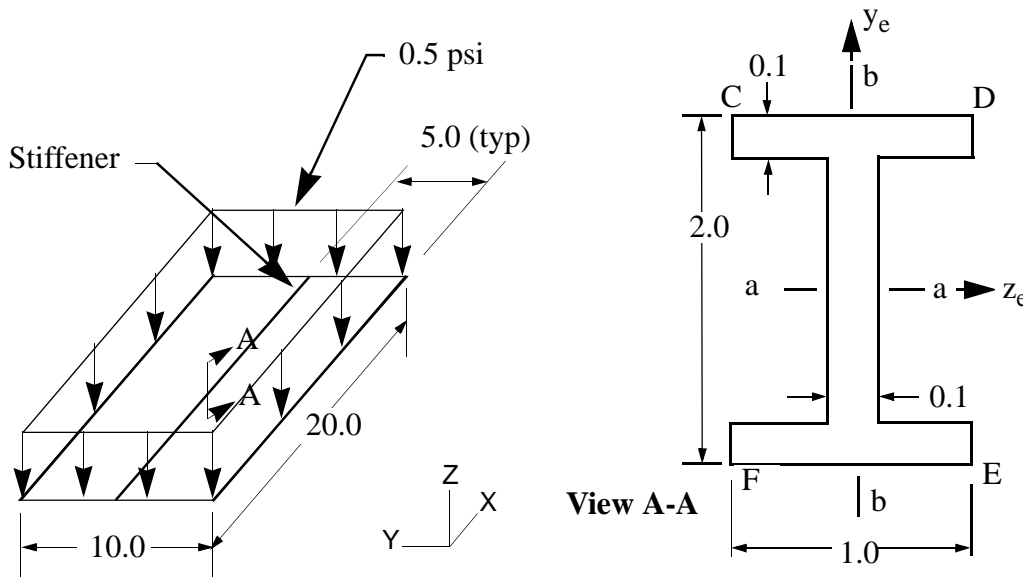


Table 3.1 - Model Properties

Elastic Modulus:	10.3E+06 psi
Poisson Ratio	0.30
Density:	0.101 lbs/in³
Plate Thickness:	0.10 in
Bar Cross-Sectional Area:	0.38 in²
I_{aa}:	0.2293 in⁴
I_{bb}:	0.0168 in⁴

J:	0.0013 in⁴
-----------	------------------------------

Suggested Exercise Steps:

- Explicitly generate a finite element representation of the stiffened plate using nodes (GRID), element connectivities, (CBAR) and a manually defined CQUAD4 element.
- Define material (MAT1) and element properties (PSHELL & PBAR).
- Verify XY-orientation and offset vectors for the bar elements.
- Define simply-supported boundary constraints (SPC1) and apply a uniform pressure load to the plate (PLOAD4).
- Use the load and boundary condition sets to define a load case (SUBCASE).
- Prepare the model for a Linear Static analysis (SOL 101).
- Generate and submit input file for MSC.Nastran.
- Review the results.

Figure 3.2 - Grid Coordinates and Element Connectivities

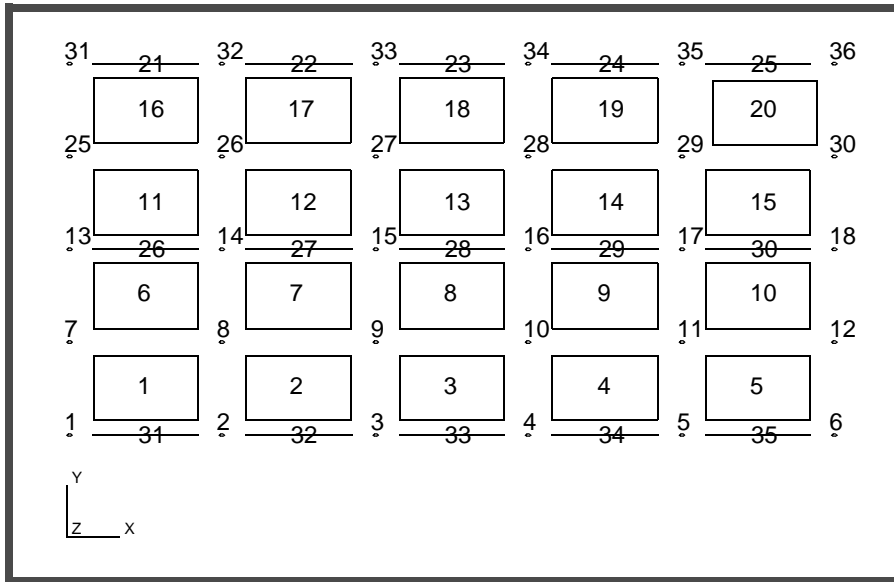
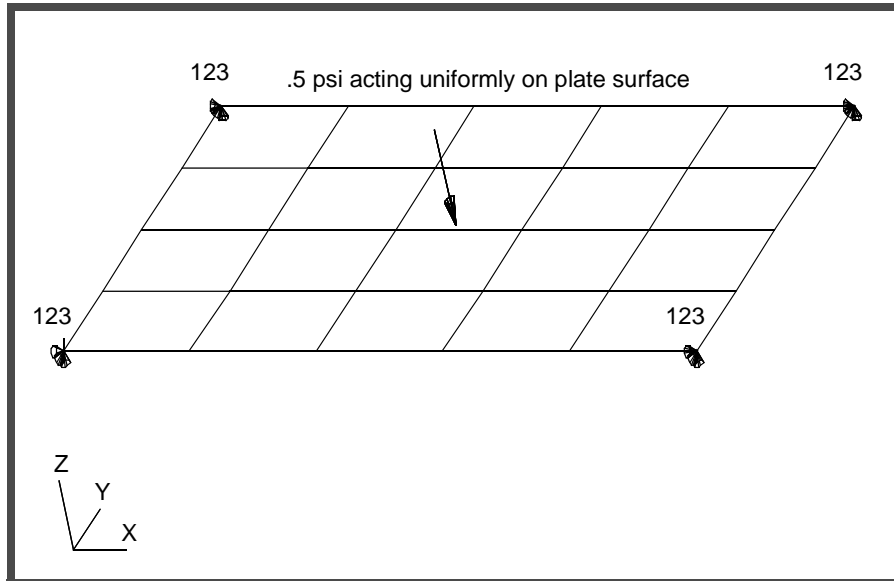


Figure 3.3 - Loads and Boundary Conditions



Exercise Procedure:

1. Users who are not utilizing MSC.Patran for generating an input file should go to Step 17, otherwise, proceed to Step 2.

2. Create a new database called **prob3.db**.

File/New...

New Database Name:

prob3

OK

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

Tolerance:

◆ **Default**

Analysis Code:

MSC/NASTRAN

Analysis Type:

Structural

OK

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

3. Create a 20x10 inch surface.

◆ **Geometry**

Action:

Create

Object:

Surface

Method:

XYZ

Vector Coordinate List:

<20, 10, 0>

Origin Coordinate List:

[0,0,0]

Apply

For clarity, turn on the **Show Parametric Direction** parameter.

Display/Geometry...

■ **Show Parametric Direction**

Apply

Cancel

4. Edit the surface by breaking it into two halves. You control how the surface is to be divided by using the **Break Direction** parameter; *Constant u Direction* corresponds to Parametric direction 1 as displayed on the surface created in the previous step.

◆ **Geometry**

Action:

Edit

Object:

Surface

Method:

Break

Option:

Parametric

Break Direction:

◆ **Constant u Direction**

Break curve:

0.5

■ **Delete Original Surfaces**

Surface List:

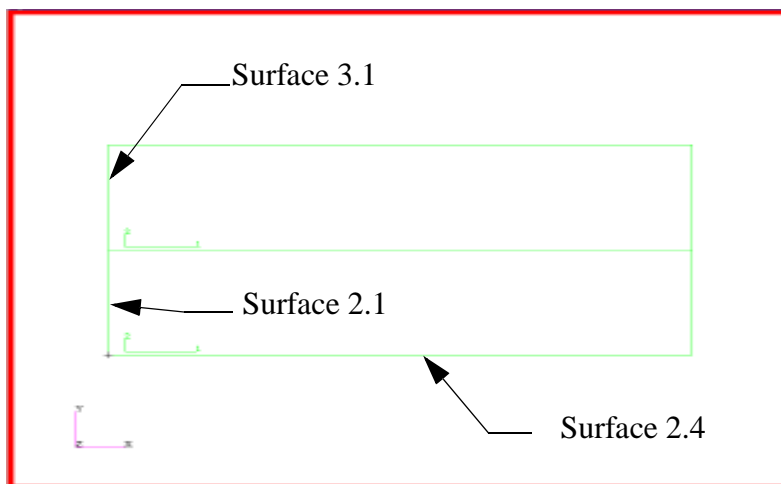
Surface 1

Apply

Answer **Yes** when the question “Do you wish to delete the original surfaces?” comes up on the screen.

Yes

Figure 3.4 - Geometry and Parametric Direction of Plates



-
5. Before you mesh the geometry model, shrink the elements by 20%; this allows you to easily visualize the element connectivities.

Display/Finite Elements...

FEM Shrink:

0.20

Apply

Cancel

To better visualize the connectivities, increase the node display size using the following toolbar icon:



Node Size

Turn off Parametric Direction to minimize the model information in your display.

Display/Geometry...

Show Parametric Directions

Apply

Cancel

6. Place mesh seeds to determine the size of the mesh.

◆ **Finite Elements**

Action:

Create

Object:

Mesh Seed

Type:

Uniform

◆ **Number of Elements**

Number =

2

Curve List:

Surface 2.1, 3.1

(Select both left edges)

Apply

Number =

5

Curve List:

Surface 2.4

(Select bottom edge)

Apply

7. Proceed with meshing the geometry model.

First, discretize the surface into Quad4 elements:

◆ **Finite Elements**

Action:

Create

Object:

Mesh

Type:

Surface

Element Topology:

Quad4

Mesher:

◆ IsoMesh

Surface List:

Surface 2, 3

(Select all surfaces)

Apply

8. To represent the stiffeners, generate bar elements along the longitudinal edges of the surfaces. There is no need to specify a Global Edge Length since the mesher will utilize existing nodes generated when you meshed the plate geometry with quad elements.

NOTE: The stiffener centroidal offsets are NOT taken into account during this step. These offsets are specified when you define the **Element Properties** for the bar elements.

◆ **Finite Elements**

Action:

Create

Object:

Mesh

Type:

Curve

Element Topology:

Bar2

Curve List:

Surface 2.4 2.2 3.2

(Select all horizontal edges)

Apply

-
9. Equivalence the model to remove duplicate nodes at common geometry boundaries.

◆ **Finite Elements**

Action:

Equivalence

Object:

All

Method:

Tolerance Cube

Apply

Refresh the screen as needed using the **Refresh Graphics** icon on the Top Menu Bar.



Refresh Graphics

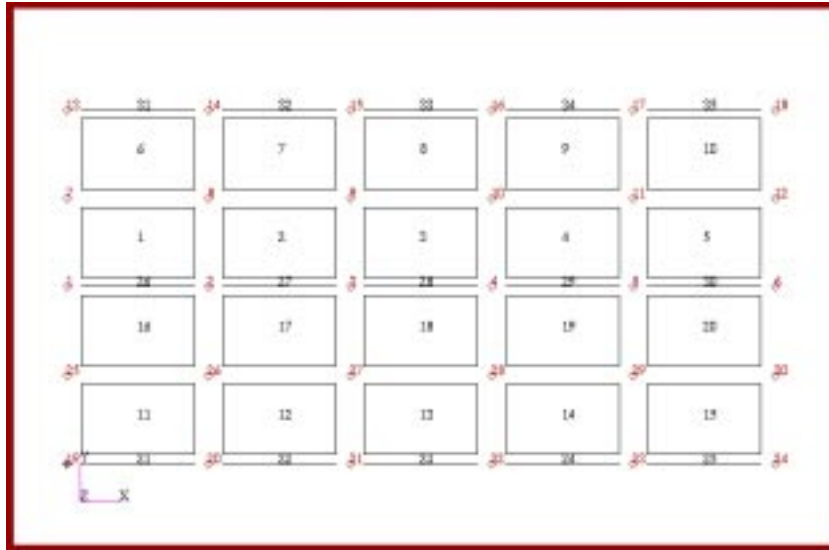
Use the Label Control Icon



to turn on node and element label



Figure 3.5 - Geometry and Meshing of Plates



- Define a material using the specified Modulus of Elasticity, Poisson Ratio & Density of the model description.

◆ **Materials**

Action:

Create

Object:

Isotropic

Method:

Manual Input

Material Name:

alum

Input Properties...

Constitutive Model:

Linear Elastic

Elastic Modulus =

10.3E6

Poisson Ratio =

0.3

OK

Apply

- Create the elements properties for the model.

- 11a. First define properties for the Quad4 elements which represent the plate.

◆ **Properties**

Action:

Dimension:

Type:

Property Set Name:

Material Name:

Thickness:
(Enter the plate thickness.)

Select Members:

(Select all surfaces.)

- 11b. Next, define properties for the Bar2 elements which represent the stiffeners. For this model, in addition to bar orientation, area, area moments of inertia, torsional constant and appropriate stress recovery coefficients, we need to define offsets (See **Hint** on page 3-3).

◆ **Properties**

Action:

Dimension:

Type:

Property Set Name:

Material Name:

Bar Orientation:

[Offset @ Node 1]:

[Offset @ Node 2]:

Area:

[Inertia 1,1]:
 (Enter Inertia about 1-1)

[Inertia 2,2]:
 (Enter Inertia about 2-2)

[Torsional Constant]:

[Y of Point C]:

[Z of Point C]:

[Y of Point D]:
 (Enter Y of Point D)

[Z of Point D]:
 (Enter Z of Point D)

[Y of Point E]:

[Z of Point E]:

[Y of Point F]:

[Z of Point F]:

Select Members:
 (Select all horizontal edges)

12. Use the **Viewing/Angles...** option to change the view. Also, erase all geometry using the **Display/Plot/Erase...** option.

Viewing/ Angles...

Method:

Angles:

Cancel

Display/Plot/Erase...

Geometry:

Erase

OK

Refresh the screen as needed using the **Refresh Graphics** icon on the Top Menu Bar.



Refresh Graphics

Also turn all label off by using Hide Label icon

- 12a. Graphically assess the orientation vectors that are required on the CBAR entries in the MSC.Nastran input file.

These vectors define the local XY plane for each bar element. Since the element property you created was applied to the geometry model instead of the analysis model, graphical display of respective attributes will appear on the geometry model by default.

In order to display attributes such as the orientation vectors on our analysis model, we must change the option in **Display/Load/BC/Elem. Props...**, since all geometry was erased from the Viewport.

Display/Load/BC/Elem. Props...

Show on FEM Only

Apply

Cancel

- 12b. Change the Action in **Properties** form to **Show**.

◆ **Properties**

Action:

Show

Existing Properties:

Definition of XY Plane

(Highlight to select)

Display Method:

Vector Plot

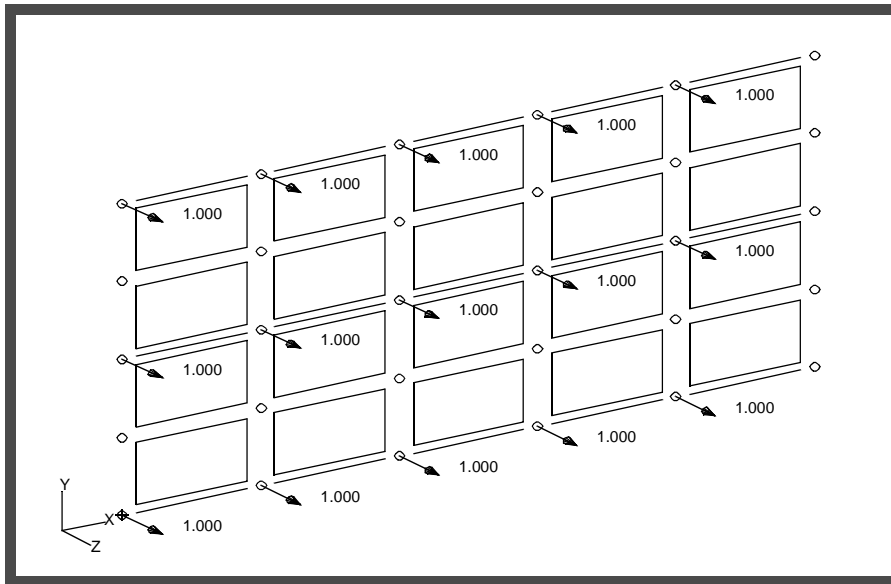
Select Group:

default_group

(Highlight to select)

Apply

Figure 3.6 - Vector Plot and Meshing of Plates



12c. Now, display the offset vector at Node 2 of each bar element.

◆ **Properties**

Action:

Show

Existing Properties:

Offset @ Node 2

(Highlight to select)

Display Method:

Vector Plot

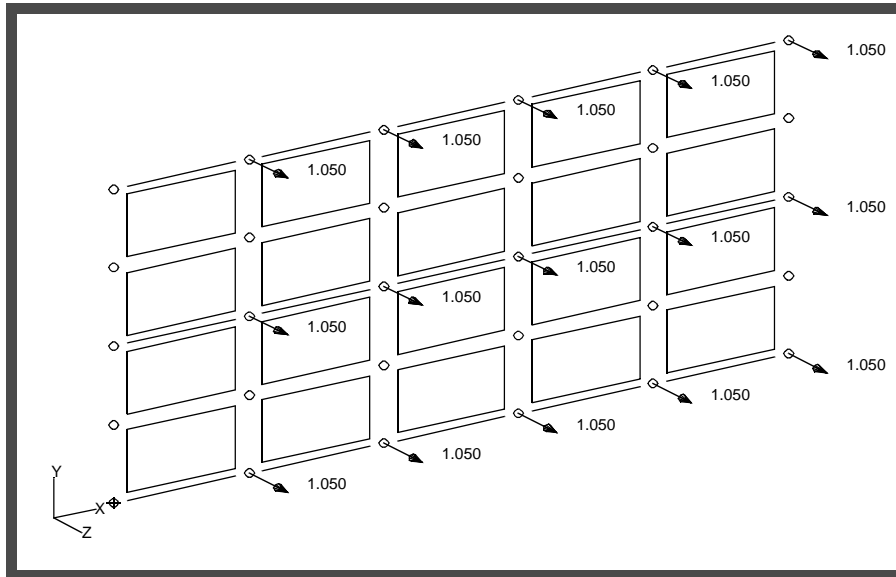
Select Group:

default_group

(Highlight to select)

Apply

Figure 3.7 - Nodal Offset and Meshing of Plates



13. Before defining loads & boundary conditions, modify your display and viewing settings as follows:

Display/Plot/Erase...

Geometry:

Plot

FEM:

Erase

OK

Display/Load/BC/Elem. Props...

Show on FEM Only

Apply

Cancel

Reset the display by selecting the **Reset Graphics** icon on the Top Menu Bar as needed before continuing.



Reset Graphics

- 13a. Define displacement constraints and apply them to the geometry model. This boundary condition represents the simply supported corners of the stiffened plate structure.

◆ **Loads/BCs**

Action:

Object:

Type:

New Set Name:

Input Data...

Translations < T1 T2 T3 >

OK

Select Application Region...

Geometry Filter:

Select Geometry Entities:

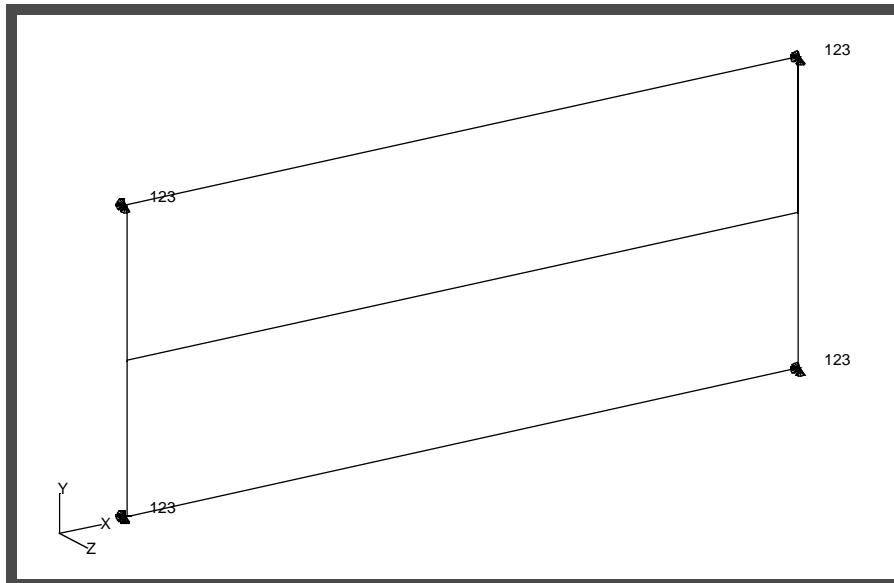
(Select the four outer corner points)

Add

OK

Apply

Figure 3.8 - Nodal Constraints of Plates



Reset the display by selecting the **Reset Graphics** icon on the Top Menu Bar.



Reset Graphics

- 13b. Apply a uniform pressure load to the surface of the plate on which the stiffeners are mounted.

◆ **Loads/BCs**

Action:

Create

Object:

Pressure

Type:

Element Uniform

New Set Name:

pressure

Target Element Type:

2D

Input Data...

Top Surf Pressure:

0.5

OK

Select Application Region...

Geometry Filter:

◆ **Geometry**

Select Surfaces or Edges:

Surface 2, 3

(Select all surfaces)

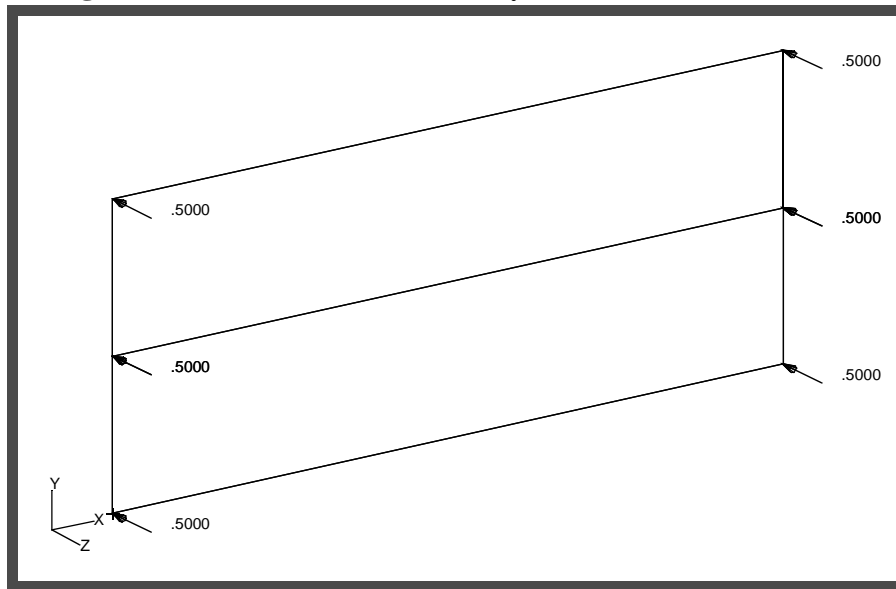
Add

OK

Apply

Note that because your pressure loads are applied to the geometry model instead of the analysis model, it may appear as if the load was not applied correctly.

Figure 3.9 - Pressures and Geometry of Plates



14. Create a new group called **fem_only**. This group will have only your analysis model entities as members.

Group/Create...

New Group Name:

fem_only

Make Current

Unpost All Other Groups

Group Contents:

Add All FEM

Apply

Cancel

15. Create a load case which references the pressure and boundary condition sets you just defined.

◆ Load Cases

Action:

Create

Load Case Name:

load_static

Load Case Type:

Static

Assign / Prioritize Loads/BCs

Select Individual Load/BCs:

Displ_simply_support

Press_pressure

(Highlight to select)

OK

Apply

16. Now you are ready to generate an input file for analysis.

Click on the **Analysis** radio button on the Top Menu Bar and complete the entries as shown here.

◆ **Analysis**

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name:

prob3

Translation Parameters...

Data Output:

XDB and Print

OK

Solution Type...

Solution Type:

◆ **Linear Static**

Solution Parameters...

■ **Database Run**

■ **Automatic Constraints**

Data Deck Echo:

Sorted

OK

OK

Subcase Select...

Subcases For Solution Sequence:

load_static

Subcases Selected:

Default

(Click to deselect)

OK

Apply

An input file called **prob3.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:

17. MSC.Nastran users can generate an input file using the data from pages 3-3 (general model description) and 3-4 (NASTRAN model element IDs). The result should be similar to the output below (**prob3.dat**):

```
ID SEMINAR,PROB3
TIME 10
SOL 101
CEND
TITLE = STIFFENED PLATE
SUBCASE 1
  SUBTITLE = load_static
  SPC = 1
  LOAD = 1
  DISPLACEMENT=ALL
  SPCFORCES=ALL
  STRESS=ALL
BEGIN BULK
PARAM, POST,0
PSHELL,1,1,.1,1,,1
CQUAD4,1,1,1,2,8,7
  =,*1,=,*1,*1,*1,*1
  =3
CQUAD4,6,1,7,8,14,13
  =,*1,=,*1,*1,*1,*1
  =3
CQUAD4,11,1,13,14,26,25
  =,*1,=,*1,*1,*1,*1
  =3
CQUAD4,16,1,25,26,32,31
  =,*1,=,*1,*1,*1,*1
  =3
PBAR    2      1      .38      .2293      .0168      .0013
        1.      -.5      1.      .5      -1.      .5      -1.      -.5
CBAR    21     2      31      32      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    22     2      32      33      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    23     2      33      34      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    24     2      34      35      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    25     2      35      36      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    26     2      13      14      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    27     2      14      15      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    28     2      15      16      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
CBAR    29     2      16      17      0.      0.      1.
        0.      0.      1.05     0.      0.      1.05
```

CBAR	30	2	17	18	0.	0.	1.	
			0.	0.	1.05	0.	0.	1.05
CBAR	31	2	1	2	0.	0.	1.	
			0.	0.	1.05	0.	0.	1.05
CBAR	32	2	2	3	0.	0.	1.	
			0.	0.	1.05	0.	0.	1.05
CBAR	33	2	3	4	0.	0.	1.	
			0.	0.	1.05	0.	0.	1.05
CBAR	34	2	4	5	0.	0.	1.	
			0.	0.	1.05	0.	0.	1.05
CBAR	35	2	5	6	0.	0.	1.	
			0.	0.	1.05	0.	0.	1.05

MAT1,1,1.03+7,,.3,.101

GRID,1,,0.,0.,0.

=,*1,=*4,==

=4

GRID,7,,0.,2.5,0.

=,*1,=*4,==

=4

GRID,13,,0.,5.,0.

=,*1,=*4,==

=4

GRID,25,,0.,7.5,0.

=,*1,=*4,==

=4

GRID,31,,0.,10.,0.

=,*1,=*4,==

=4

SPC1,1,123,1,6,31,36

PLOAD4,1,1,-.5,,,,,THRU,20

ENDDATA

Submitting the Input File for Analysis:

18. Submit the input file to MSC.Nastran for analysis.
 - 18a. To submit the MSC.Patran **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob3.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
 - 18b. To submit the MSC.Nastran **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob3.dat scr=yes**. Monitor the run using the UNIX **ps** command.
19. When the run is completed, edit the **prob3.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.
 - 19a. While still editing **prob3.f06**, search for the word:

D I S P L A C E (spaces are necessary).

What are the components of the displacement vector for GRID 16 (translation only)?
(Please check Figure 3-5 for exact location. Your ID may be different because of the meshing sequence)

disp X = _____
disp Y = _____
disp Z = _____

Search for the word:

S T R E S S (spaces are necessary)

What is the axial stress for CBAR 28?
(Please check Figure 3-5 for exact location. Your ID may be different because of the meshing sequence)

axial stress = _____

Search for the word:

Q U A D (spaces are necessary)

What are the centroidal Von Mises stresses for

CQUAD4 13?

-(thk/2): stress = _____
+(thk/2): stress = _____

Comparison of Results:

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

21. **MSC.Nastran Users have finished this exercise. MSC.Patran Users should proceed to the next step.**
22. Proceed with the Reverse Translation process, that is, attaching the **prob3.xdb** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	Attach XDB
<i>Object:</i>	Result Entities
<i>Method:</i>	Local
Select Results File...	
<i>Available Files:</i>	prob3.xdb
OK	
Apply	

23. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.
- 23a. Select **Fringe** to view different results with color spectrum analysis.

◆ **Results**

<i>Action:</i>	Create
<i>Object:</i>	Fringe

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



Select Results

<i>Select Result Case(s):</i>	load_static, Static Subcase
<i>Select Fringe Result:</i>	Displacements, Translational
<i>Quantity:</i>	Z Component

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



Display Attributes

Style:

Continuous

Style:



Apply

23b. 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 Result Case(s):

load_static, Static Subcase

Select Deformation Result:

Displacements, Translational

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



Display Attributes

Show Undeformed

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)*, *Fringe Result*, and *Deformation Result*.

Quit MSC.Patran when you have completed this exercise.

