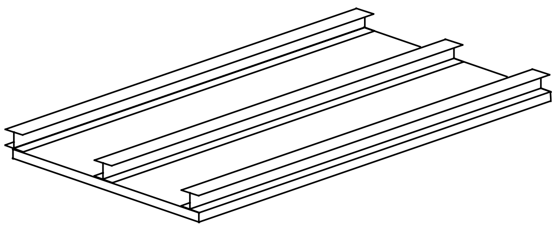
WORKSHOP PROBLEM 3

Linear Static Analysis of a Simply-Supported Stiffened Plate



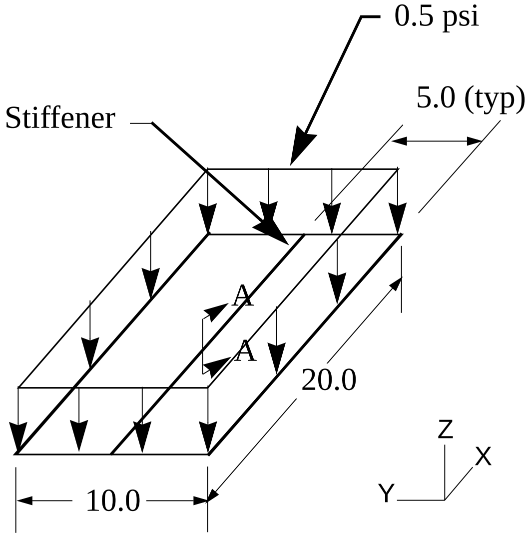
Objectives:

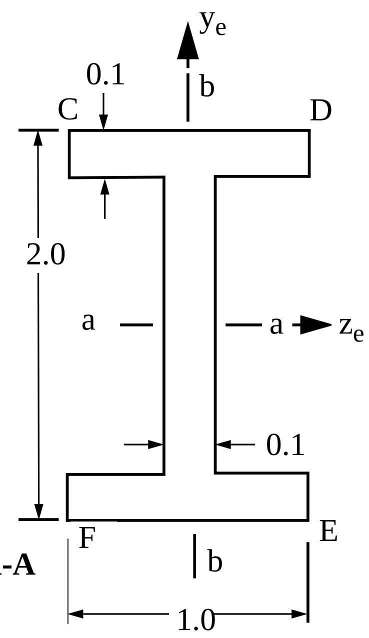
- Create a MSC/NASTRAN analysis model comprised of CQUAD4 & CBAR elements.
- Prepare a MSC/NASTRAN input file for a Linear Static analysis.
- Visualize analysis results.

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 3-2). 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.





View A-A

Table 3.1

Elastic Modulus:	**10.3E6 psi**
Poisson Ratio	**0.3**
Density:	**0.101 lbs/in^3**
Plate Thickness:	**0.1 in**
Bar cross sectional area:	**0.38 in^2**
I_{aa}:	**0.2293 in^4**
I_{bb}:	**0.0168 in^4**
J:	**0.0013 in^4**

Suggested Exercise Steps:

- Explicitly generate a finite element representation of the stiffened plate using nodes (GRID), element connectivies, (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 loadcase (SUBCASE).
- Prepare the model for a Linear Static analysis (SOL 101).
- Generate and submit input file for MSC/NASTRAN.
- Review the results.

Figure 3.1

Grid Coordinates and Element Connectivities

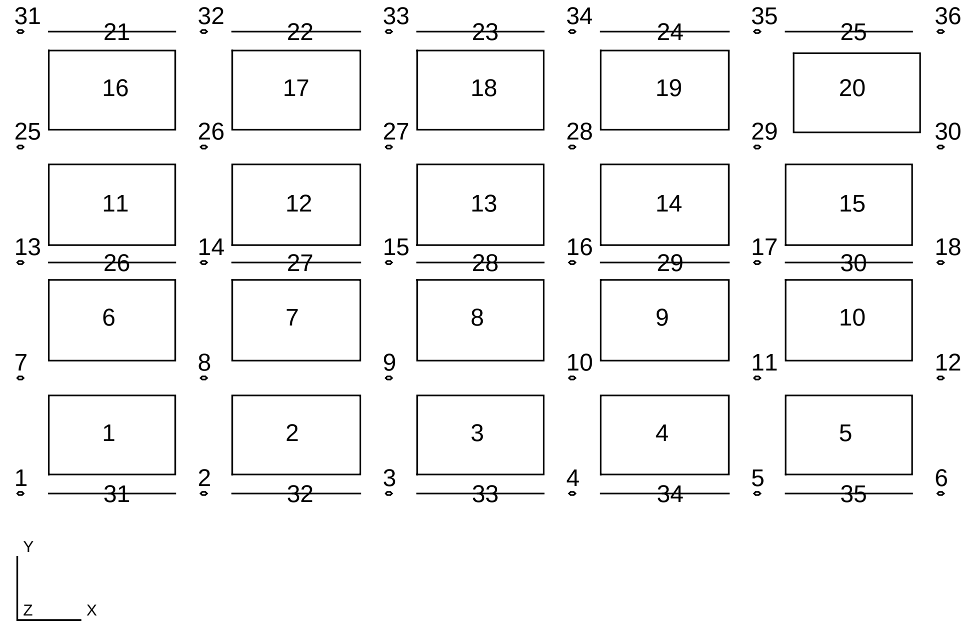
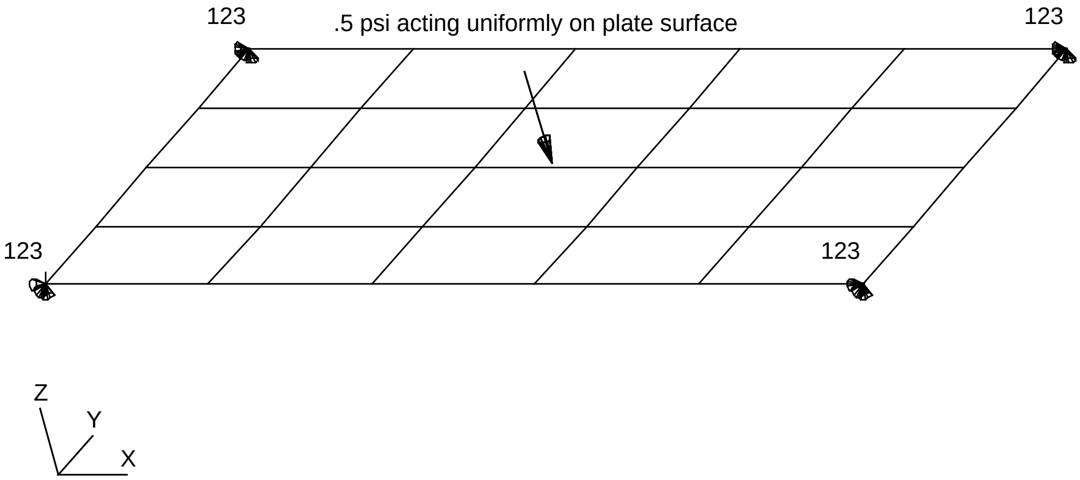


Figure 3.2

Loads and Boundary Conditions



```
ID SEMINAR,PROB3
```

```
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 utilitizing 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

3. Create a 20x10 inch surface.

◆ **Geometry**

Action: **Create**

Object: **Surface**

Method: **XYZ**

Vector Coordinate List: **<20, 10, 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**

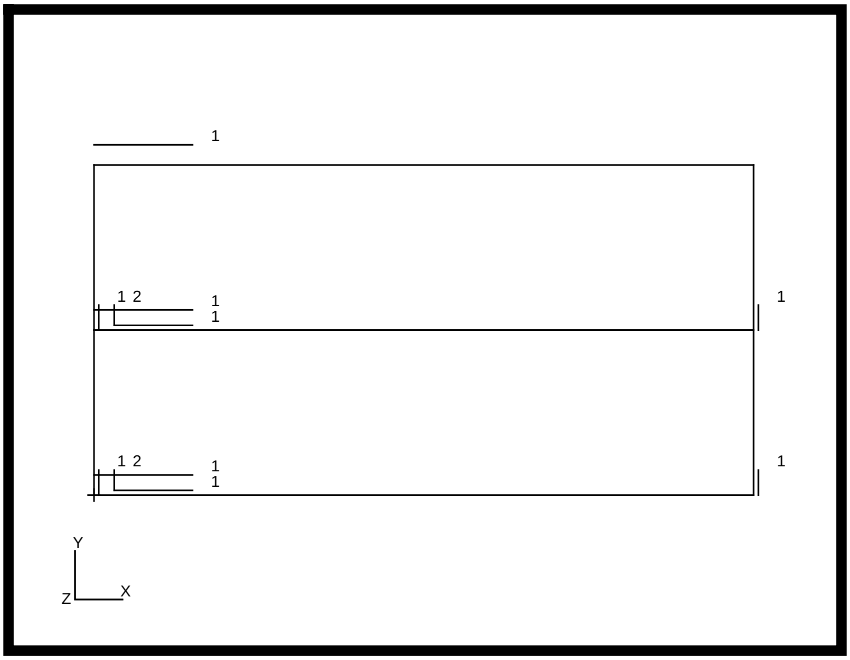
■ **Auto Execute**

Surface List:	**Surface 1**

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

Yes

Figure 3.3 - Geometry and Parametric Direction of Plates



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

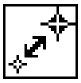
Display/Finite Elements...

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 2.2 2.4 3.2** (select all horizontal edges)

Apply

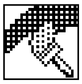
9. Equivalence the model to remove duplicate nodes at common surface edges.

◆ **Finite Elements**

Action:	**Equivalence**
Object:	**All**
Method:	**Tolerance Cube**

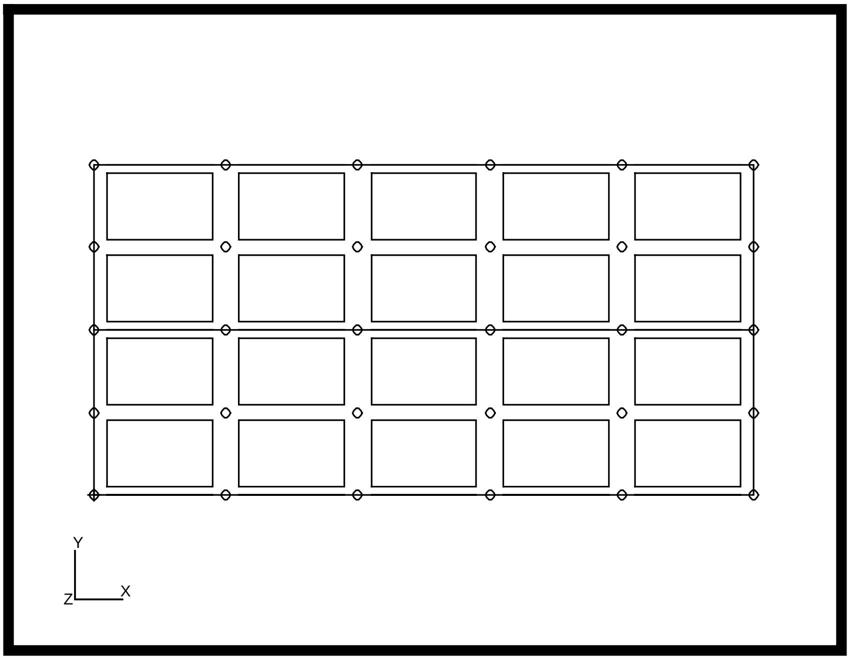
Apply

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



Refresh Graphics

Figure 3.4 - Geometry and Meshing of Plates



10. 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**
Density =	**0.101**

Apply

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

Cancel

11. Create the elements properties for the model.

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

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name:	**plate**

Input Properties ...

Material Name:	**m:alum**
Thickness:	**???** (Enter the plate thickness)

OK

Select Members: **Surface 2 3**
(select all surfaces)

Add

Apply

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:	**Create**
Dimension:	**1D**
Type:	**Beam**
Property Set Name:	**stiffener**

Input Properties ...

Material Name:	**m:alum**
Bar Orientation:	**<0., 0., 1.>**
[Offset @ Node 1]:	**<0. 0. 1.05>**
[Offset @ Node 2]:	**<0. 0. 1.05>**
Area:	**0.38**
[Inertia 1,1]:	**???** (Enter Inertia about 1-1)
[Inertia 2,2]:	**???** (Enter Inertia about 2-2)
[Torsional Constant]:	**0.0013**
[Y of Point C]:	**1.**
[Z of Point C]:	**-0.5**
[Y of Point D]:	**???** (Enter Y of Point D)
[Z of Point D]:	**???** (Enter Z of Point D)
[Y of Point E]:	**-1.**

[Z of Point E]: **0.5**

[Y of Point F]: **-1.**

[Z of Point F]: **-0.5**

OK

Select Members: **Surface 2.2 2.4 3.2**
(select all horizontal edges)

Add

Apply

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

Viewing/ Angles...

Method: ◆ **Model Absolute**

Angles: **23.0, 34.0, 0.0**

Apply

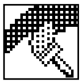
Cancel

Display/Plot/Erase...

Erase All Geometry

OK

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

 **Refresh Graphics**

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

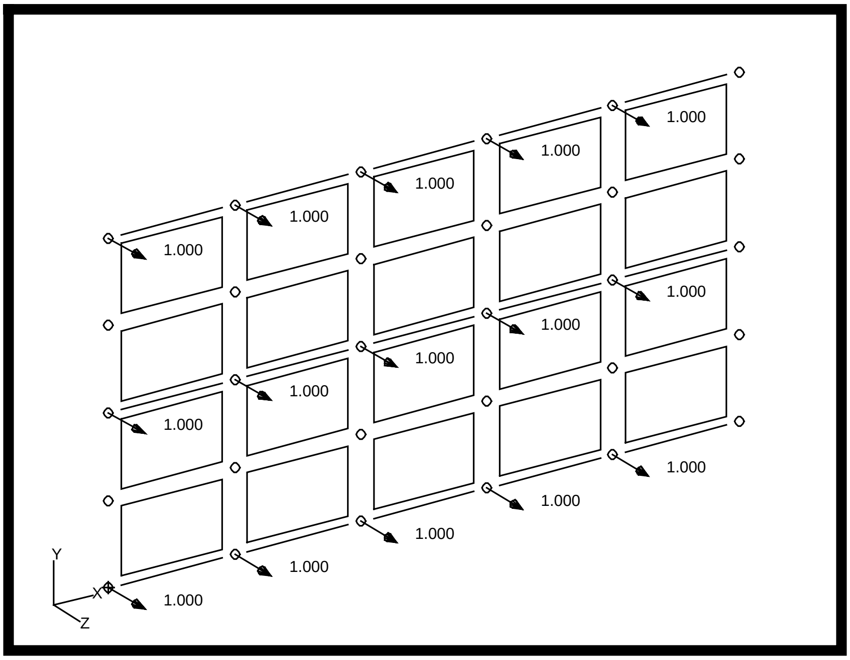
12b. Change the Action in **Properties** form to **Show**.

◆ **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 3.5 - Vector Plot and Meshing of Plates



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

◆ **Properties**

Action: **Show**

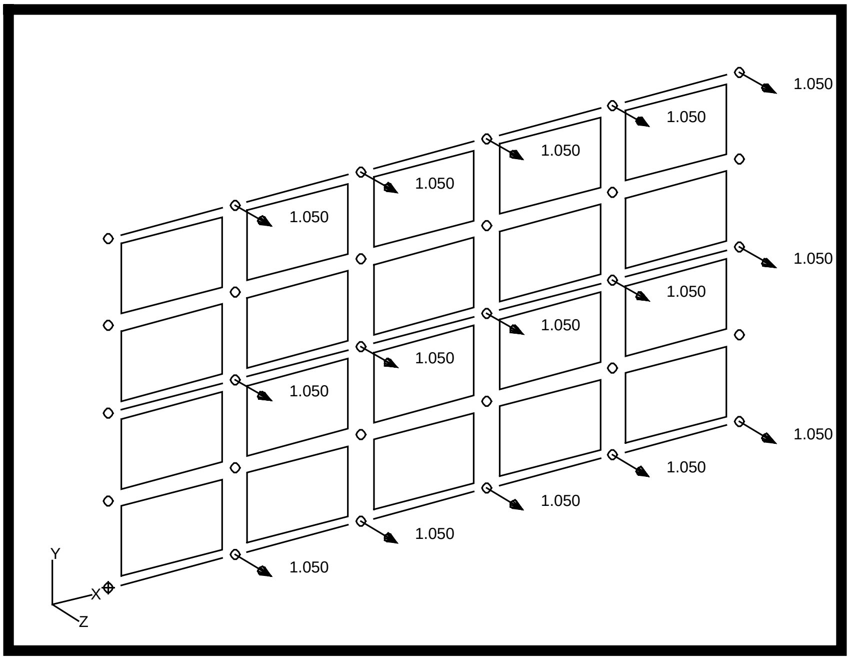
Existing Properties: **Offset @ Node 2**
(highlight to select)

Display Method: **Vector Plot**

Select Group: **default_group**
(highlight to select)

Apply

Figure 3.6 - Nodal Offset and Meshing of Plates



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

Display/Plot/Erase...

Plot All Posted Geometry

Erase All FEM

OK

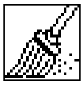
Display/Load/BC/Elem. Props...

❑ **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:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**simply_support**

Input Data...

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

OK

Select Application Region...

Geometry Filter:	◆ **Geometry**
Select Geometry Entities:	**Point 1 2 3 4**

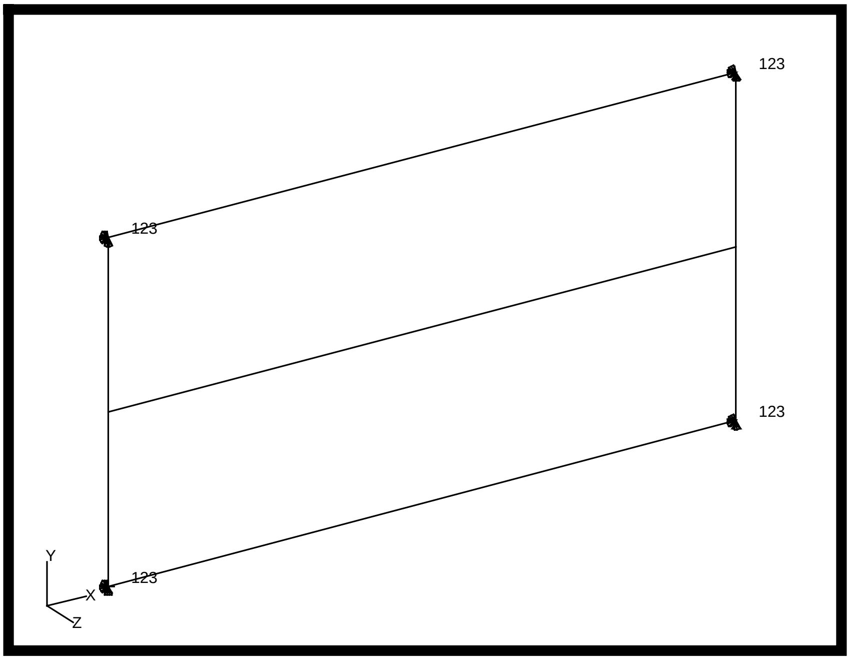
(select the four outer corner points)

Add

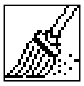
OK

Apply

Figure 3.7 - 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)

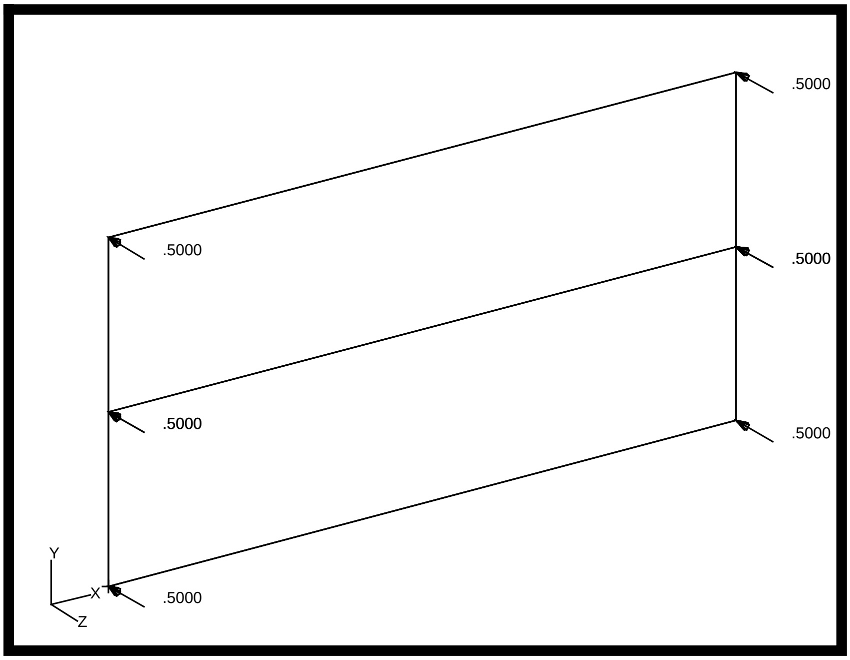
Add

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 3.8 - 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 loadcase 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 Loads/BCs to Add to Spreadsheet:	**Displ_simply_support** **Press_pressure**
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...	
OUTPUT2 Format:	**Binary**
MSC/NASTRAN Version:	**???** Set accordingly, here it is 70.
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. MSC/PATRAN users should now proceed to **Step 18**.

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
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
```

Generating an input file (cont.):

```
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 MSC/NASTRAN and MSC/PATRAN USERS:

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 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)?

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?

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:

```
                                     D I S P L A C E M E N T   V E C T O R
   POINT ID.   TYPE          T1             T2             T3             R1             R2             R3
       ...
         15      G     -1.476435E-06   6.064756E-19  -6.628885E-02  -9.887924E-17  -7.129618E-07   1.301043E-18
         16      G      1.476435E-06   4.320388E-19  -6.628885E-02  -8.207097E-17   7.129618E-07  -9.923630E-18
         17      G      5.591691E-06   1.752623E-19  -6.629492E-02  -4.008553E-17   2.532921E-06  -1.007822E-17
       ...
                             S T R E S S E S   I N   B A R   E L E M E N T S       ( C B A R )
   ELEMENT     SA1           SA2           SA3           SA4          AXIAL         SA-MAX        SA-MIN     M.S.-T
     ID.       SB1           SB2           SB3           SB4          STRESS        SB-MAX        SB-MIN     M.S.-C
    ...
    27    6.330274E+00   6.330274E+00  -6.330274E+00  -6.330274E+00   1.551750E+01   2.184777E+01   9.187223E+00
          3.042516E+00   3.042516E+00  -3.042516E+00  -3.042516E+00                  1.856001E+01   1.247498E+01
    28    3.671753E+00   3.671753E+00  -3.671753E+00  -3.671753E+00   1.145898E+01   1.513073E+01   7.787226E+00
          3.671753E+00   3.671753E+00  -3.671753E+00  -3.671753E+00                  1.513073E+01   7.787226E+00
    29    3.042516E+00   3.042516E+00  -3.042516E+00  -3.042516E+00   1.551750E+01   1.856001E+01   1.247498E+01
          6.330274E+00   6.330274E+00  -6.330274E+00  -6.330274E+00                  2.184777E+01   9.187223E+00
    ...
                        S T R E S S E S   I N   Q U A D R I L A T E R A L   E L E M E N T S   ( Q U A D 4 )

ELEMENT        FIBRE        STRESSES IN ELEMENT COORD SYSTEM        PRINCIPAL STRESSES (ZERO SHEAR)
ID  GRID-ID   DISTANCE      NORMAL-X     NORMAL-Y     SHEAR-XY        ANGLE      MAJOR         MINOR        VON MISES
  ...
13  CEN/4 -5.000000E-02  9.104982E+02  3.264549E+03 -1.470901E-10 -90.0000  3.264549E+03  9.104982E+02  2.917862E+03
           5.000000E-02 -8.685194E+02 -3.222543E+03  1.413458E-10  0.0000  -8.685194E+02 -3.222543E+03  2.887952E+03
       15 -5.000000E-02  9.943295E+02  3.289698E+03 -1.470901E-10 -90.0000  3.289698E+03  9.943295E+02  2.922287E+03
           5.000000E-02 -9.679219E+02 -3.252364E+03  1.413458E-10  0.0000  -9.679219E+02 -3.252364E+03  2.892526E+03
       16 -5.000000E-02  9.943295E+02  3.289698E+03 -1.470901E-10 -90.0000  3.289698E+03  9.943295E+02  2.922287E+03
           5.000000E-02 -9.679219E+02 -3.252364E+03  1.413458E-10  0.0000  -9.679219E+02 -3.252364E+03  2.892526E+03
       28 -5.000000E-02  8.266670E+02  3.239399E+03 -1.471567E-10 -90.0000  3.239399E+03  8.266670E+02  2.915335E+03
           5.000000E-02 -7.691169E+02 -3.192722E+03  1.414124E-10  0.0000  -7.691169E+02 -3.192722E+03  2.886077E+03
       27 -5.000000E-02  8.266670E+02  3.239399E+03 -1.470901E-10 -90.0000  3.239399E+03  8.266670E+02  2.915335E+03
..   ...
```

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, importing the **prob3.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:	**prob3.op2**
OK	
Apply	

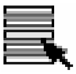
23. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

23a. Select **Fringe** to view different results with color spectrum analysis.

◆ **Results**

Action:	**Create**
Object:	**Fringe**

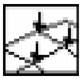
To select results, click on the **Select Results** icon.



Select Results

Select Result Case(s):	**LOAD_STATIC, Static Subcase**
Select Fringe Result:	**Displacement, Translational**
Quantity:	**Z Component**

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

 **Display Attributes**

Style:	**Continuous**
Style:	

Apply

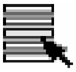
23b. Select **Deformation** to view physical changes of the model.

.

◆ **Results**

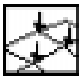
Action:	**Create**
Object:	**Deformation**

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

 **Select Results**

Select Result Case(s):	**LOAD_STATIC, Static Subcase**
Select Deformation Result:	**Displacement, Translational**

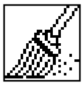
To change the Display Attributes, click on the **Display Attributes** icon.

 **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.