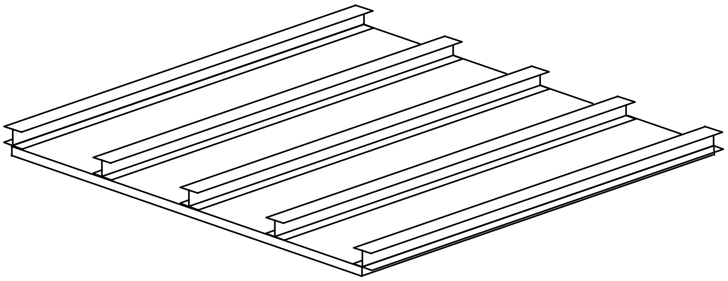
LESSON 10

Linear Static Analysis of a Simply-Supported Stiffened Plate



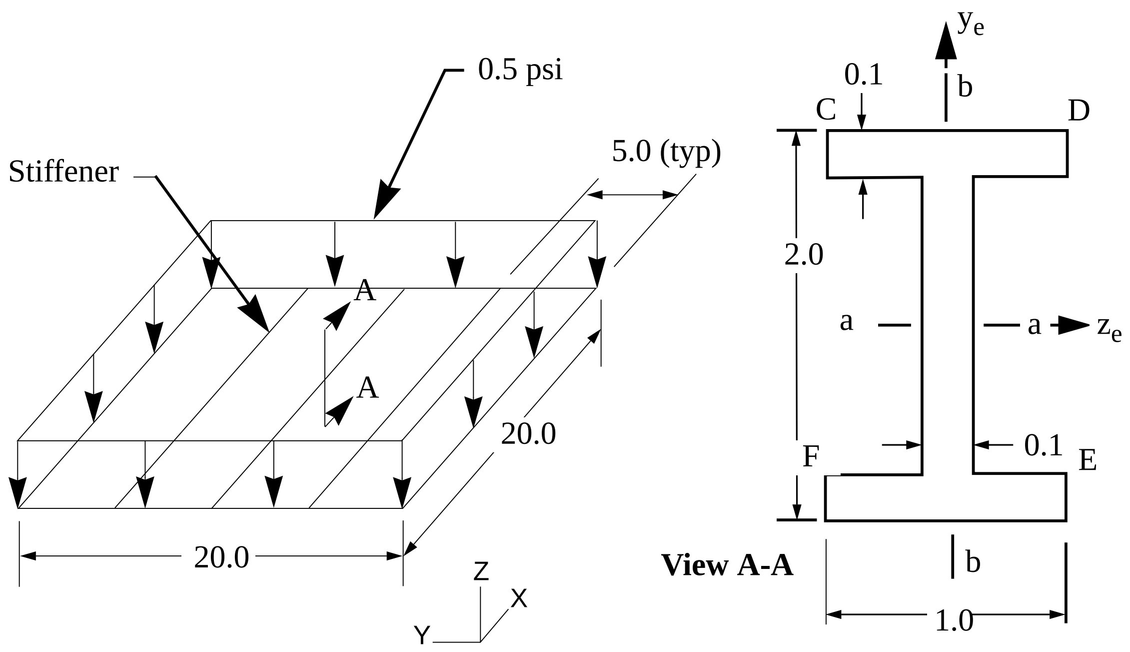
Objectives:

- Create a geometric representation of a stiffened plate.
- Use the geometry model to define 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 10-1. The plate is 0.1 inches thick therefore thin-shell theory applies. I-beam stiffeners are mounted as shown. The structure is simply supported on its four corners and a uniform pressure of 0.5 psi is applied to the surface of the 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.



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:

- Open a new database.
- Define a geometric representation of the stiffened plate using a surface.
- Define an analysis model by meshing the geometry model with shell (CQUAD4) and bar (CBAR) elements.
- Define material (MAT1) and element properties (PSHELL and 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 and PARAMs).
- Generate and submit input file for MSC/NASTRAN.
- Post-process results.
- Quit MSC/PATRAN.

Exercise Procedure:

1. Create a new database called **lesson10.db**.

File/New Database...

New Database Name:	**lesson10**
OK	

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

Tolerance:	◆ **Default**
Analysis Code:	**MSC/NASTRAN**
Analysis Type:	**Structural**
OK	

2. Create a 20x20 inch surface.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinates List:	**< 20 20 0 >**

❑ **Auto Execute**

Apply

2a. For clarity, turn on the **Show Parametric Direction**. Use the **Display/Geometry...** option.

Display/Geometry...

■ **Show Parametric Direction**

Apply

Cancel

3. Edit the surface by breaking it into two halves. You control how the surface is to be divided by the Break Direction Parameter; *Constant u Direction* corresponds to Parametric direction 1 as displayed on the surface created in Step 2.

◆ **Geometry**

Action:	**Edit**
Object:	**Surface**
Method:	**Break**
Option:	**Parametric**
Break Direction:	◆ **Constant u Direction**
Break curve:	**0.5**

■ **Delete Original Surfaces**

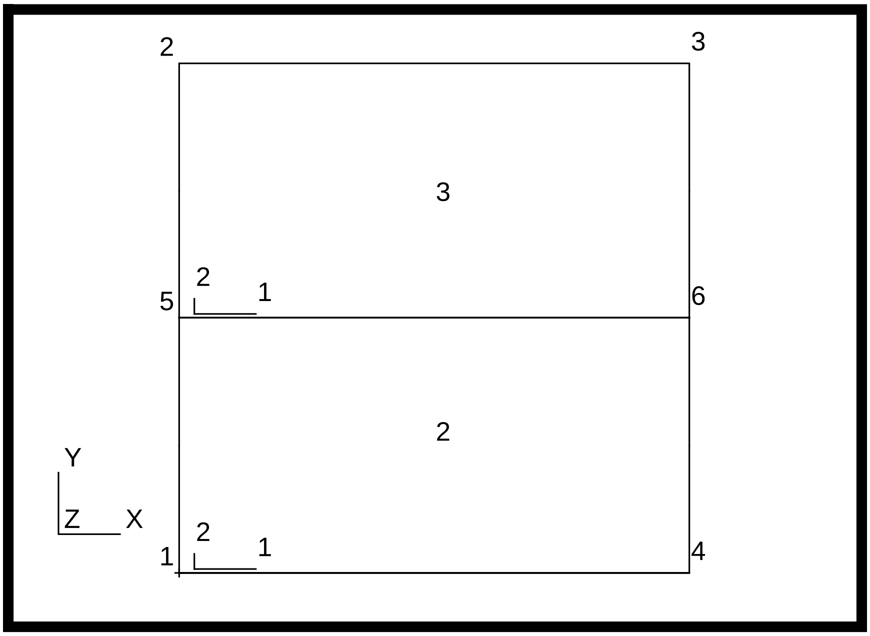
❑ **Auto Execute**

Surface List:	**Surface 1**

Apply

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

After this step, your display should show the following:



3a. Repeat the last operation to break the two new surfaces to yield a total of four new surfaces, each having the same dimensions as the other.

Recall that you may either use the keyboard to specify the ids of the desired surfaces explicitly in the *Surface List* databox using the proper picklist syntax, OR

You may specify the desired surface with your mouse by first clicking in the *Surface List* databox and then clicking desired surface in the **default_viewport**.

Surface List:	**Surface 2**

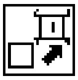
Apply

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

Surface List:	**Surface 3**

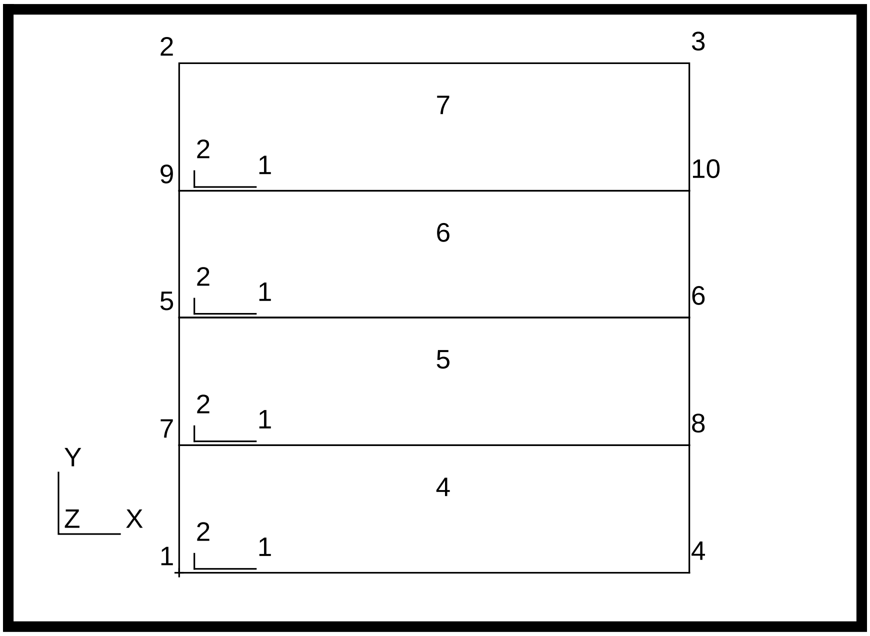
Apply

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



Show Labels

The completed geometry model should appear as follows:



4. For clarity, before you mesh the geometry model, shrink the elements by 20%; this allows us to easily assess the element connectivities.

Display/Finite Element...

FEM Shrink: **0.20**

4a. To better visualize the connectivities, increase the node display size using **Display/Finite Element...**. Deactivate the labels for the surfaces and the points to minimize the model information in your display.

Node Sizes: **5**

Hide All FEM Labels

Apply

Cancel

Display/Geometry...

❑ **Show Parametric Direction**

Show All Geometry Labels

Colors and Labels:

Point: ❑ **Label**

Surface: ❑ **Label**

Apply

Cancel

5. Proceed with meshing the geometry model.

5a. First, discretize the surface into quad4 elements:

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Surface**
Global Edge Length:	**2**
Element Topology:	**Quad4**
Mesher:	◆ **IsoMesh**
Surface List:	**Surface 4:7**

Apply

5b. 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 the discretization step. These offsets are specified when you define the **Element Properties** for the bar elements.

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Element Topology:	**Bar2**
Curve List:	**Surface 4.4 4:7.2**

Note: Your curve list may be different since there are edges from two different surfaces that occupy the same location.

Apply

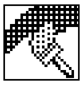
6. Equivalence the model to remove duplicate nodes at common surface edges.

◆ **Finite Elements**

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

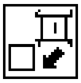
Apply

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



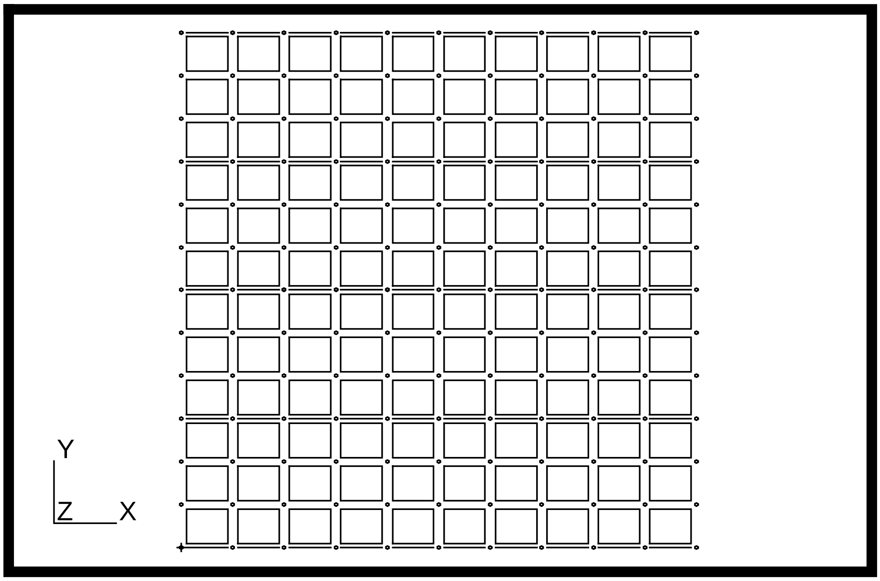
Refresh Graphics

For clarity, hide the entity labels by selecting the **Hide Labels** icon on the **Top Menu Bar**.



Hide Labels

The completed model with all entity labels hidden should appear as follows:



7. Define a material using the specified Modulus of Elasticity, Poisson ratio & density.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**alum**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**10.3E6**
Poisson Ratio =	**.3**
Density =	**.101**

Apply

The *Current Constitutive Models* form should appear as below:

Linear Elastic - [,,,,] - [Active]

Cancel

8. Reference the material you just defined when you specify element properties for your analysis model.

◆ **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 4:7**

Add

Apply

9. First define properties for the quad4 elements which represent the **plate**.

9a. 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 10-3).

Action:	**Create**	
Object:	**1D**	
Method:	**Beam**	
Property Set Name:	**bar**	
Input Properties...		
Material Name:	**m:alum**	
(Use scroll bar on the right)		
Bar Orientation:	**<0. 0. 1.>**	
[Offset @ Node 1]	**<0. 0. 1.05>**	
[Offset @ Node 2]	**<0. 0. 1.05>**	
(Use scroll bar on the right)		
Area:	**0.38**	
[Inertia 1,1]	**???**	*(Enter Inertia about 1-1)*
[Inertia 2,2]	**???**	*(Enter Inertia about 2-2)*
(Use scroll bar on the right)		
[Torsional Constant]	**0.0013**	
(Use scroll bar on the right)		
[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 4.4 4:7.2**

Add

Apply

10. 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

10a. 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. But first, you may wish to enable the bar element labels for clarity.

Display/Finite Elements...

Bar: ■ **Label**

Apply

Cancel

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

■ **Show on FEM Only**

Apply

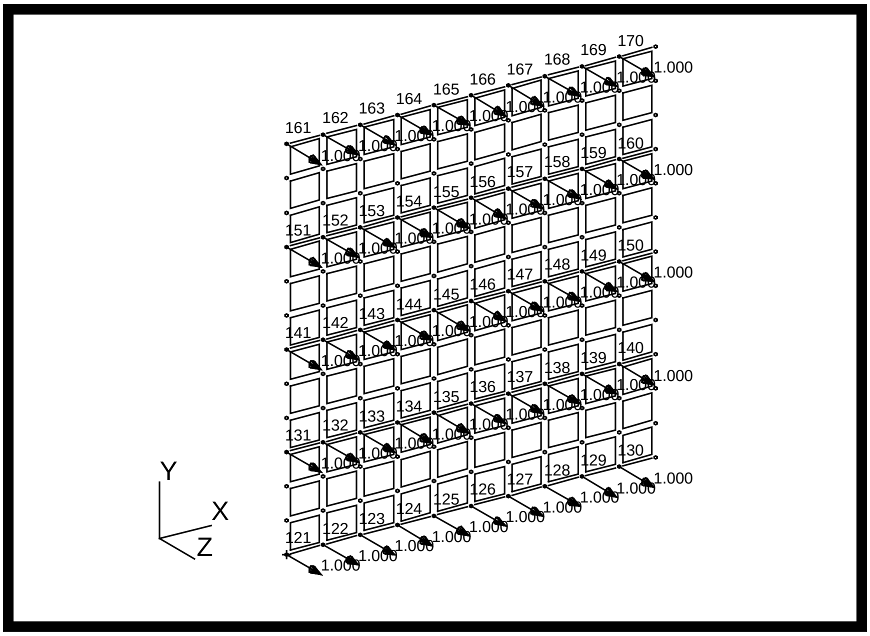
Cancel

10b. Change the Action in **Properties** form to **Show**.

Action:	**Show**
Existing Properties:	**Definition of XY Plane**
Display Method:	**Vector Plot**
Select Group:	**default_group**

Apply

The display in the viewport should appear as follows:

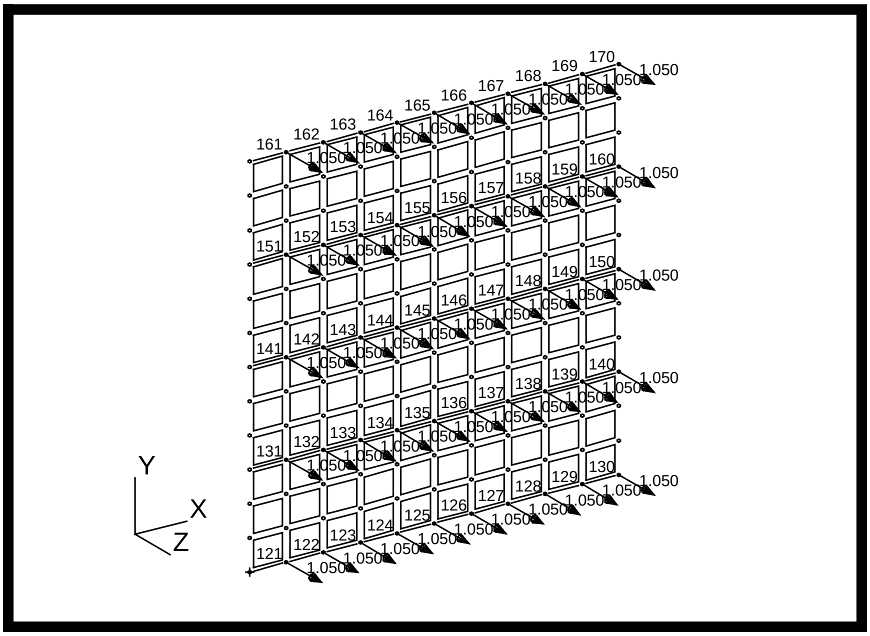


10c. Now, display the offset vector at Node 2 of each bar element.

Action:	**Show**
Existing Properties:	**Offset @ Node 2**
Display Method:	**Vector Plot**
Select Group:	**default_group**

Apply

Your display should appear as follows:



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

Display/Entity Color/Label/Render

Point:	■ **Label**
Surface:	■ **Label**
Bar:	❑ **Label**

Apply

Cancel

Display/Plot/Erase

Plot All Posted Geometry

Erase All FEM

OK

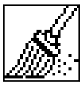
Display/Load/BC/Elem. Props

❑ **Show on FEM Only**

Apply

Cancel

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

 **Reset Graphics**

11a. Define displacement constraints and apply it to the geometry model. This boundary condition represents the simply supported corners of the stiffened plate structure.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name:	**simply_support**

Input Data...

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

OK

Select Application Region...

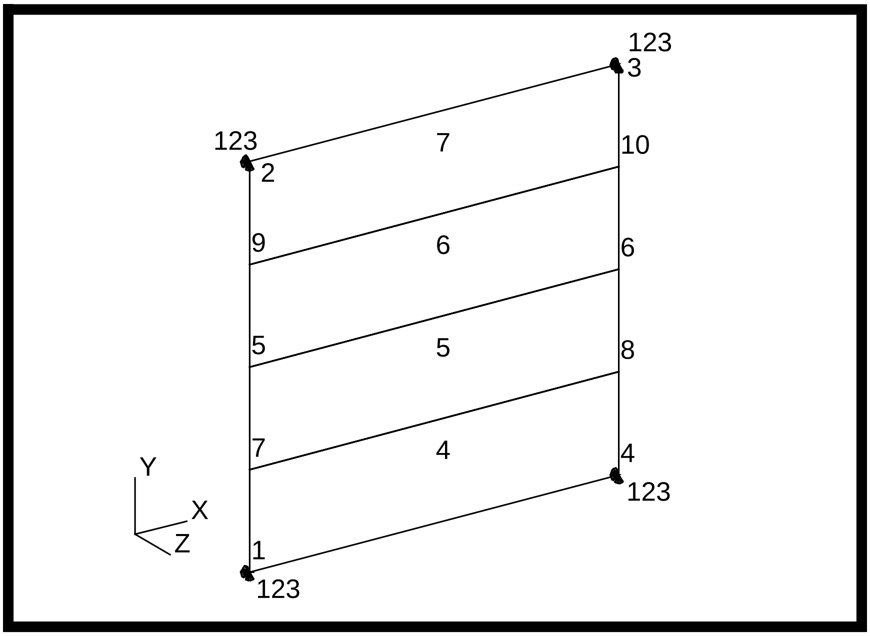
Geometry Filter:	◆ **Geometry**
Select Geometry Entities:	**Point 1:4**

Add

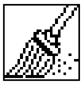
OK

Apply

Your display should appear as follows:



Reset the display by selecting the broom icon on the Top Menu Bar.

 **Reset Graphics**

11b. Apply a uniform pressure load to the surface of the plate on which the stiffeners are mounted.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Pressure**
Method:	**Element Uniform**
New Set Name:	**pressure**
Target Element Type:	**2D**

Input Data...

Top Surf Pressure:	**0.5**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

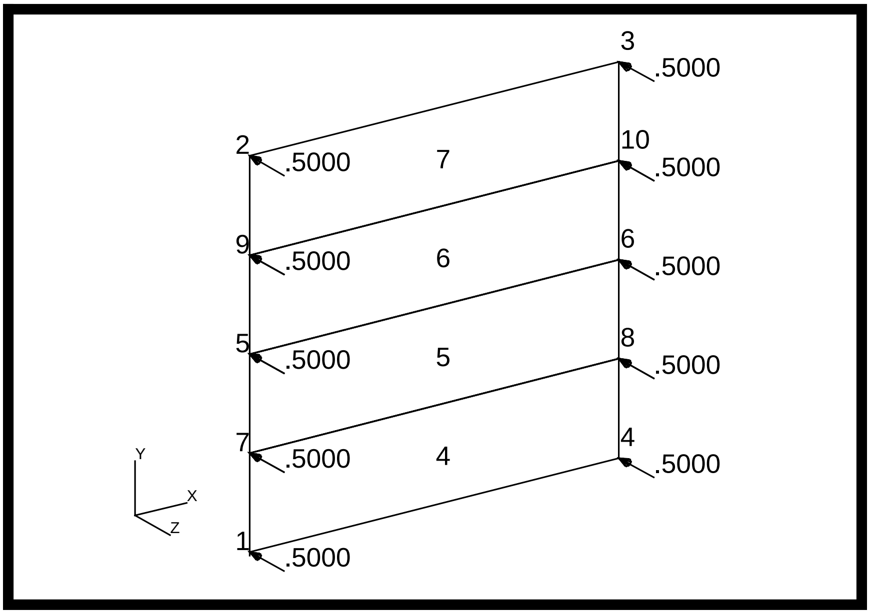
Select Geometry Entities: **Surface 4:7**

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. The applied pressure should appear as follows:



12. 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

12a. Once again, enable the **Show on FEM only** button in **Display/Load/BC/Elem Props...**.

12b. For clarity, you may to wish disable the LBC/El. Prop. Values display for your load & boundary condition sets.

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

❑ **Show LBC/El. Prop. Values**

Apply

Cancel

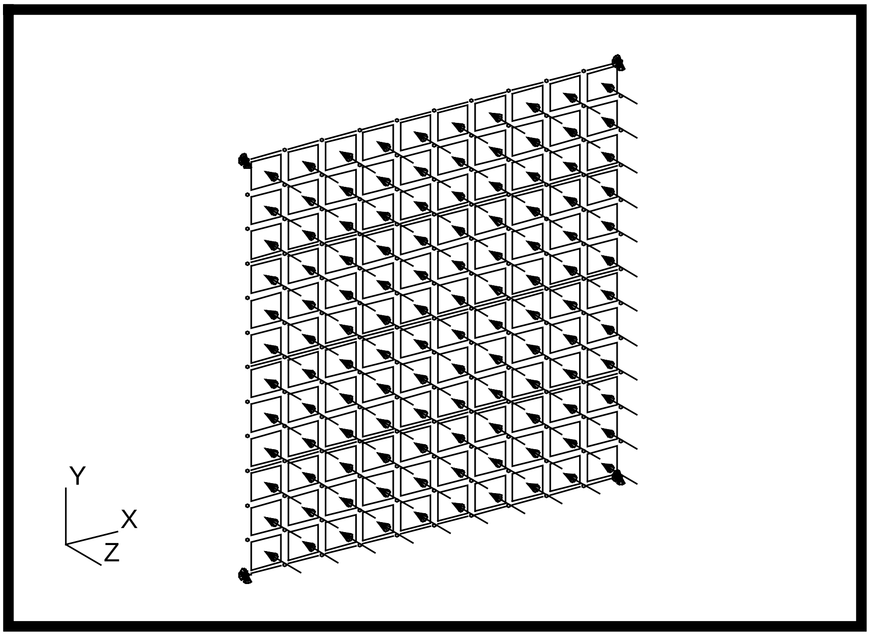
To verify the LBCs on the analysis model,

◆ **Loads/BCs**

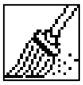
Action:	**Plot Markers**
Assigned Load/BCs Sets:	**Disp_simply_support** **Press_pressure**
Select Groups:	**fem_only**

Apply

Your model should appear as follows:



Reset the display by selecting the broom icon on the Top Menu Bar.

 **Reset Graphics**

13. Create a loadcase which references the pressure and boundary condition sets which you defined.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**load_static**
Load Case Type:	**Static**
Assign/Prioritize Loads/BCs	
*(Click each selection until all Loads/BCs have one entry in the spreadsheet)**	**Disp_simply_support** **Press_pressure**

*** <u>REMINDER:</u>** Make sure that the LBC Scale Factor column shows the proper value for each entry.

OK

Apply

14. 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:	**plate**

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**
Wt.- Mass Conversion =	**0.00259**
	(for English unit)

OK

OK

Subcase Select...

Subcases For Solution Sequence: **load_static**

Subcases Selected: **Default** *(Click to deselect)*

OK

Apply

An input file called **`plate.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.

15. If all is well, you will then submit the input file to MSC/NASTRAN for analysis. To do this, find an available xterm window and at the prompt enter:

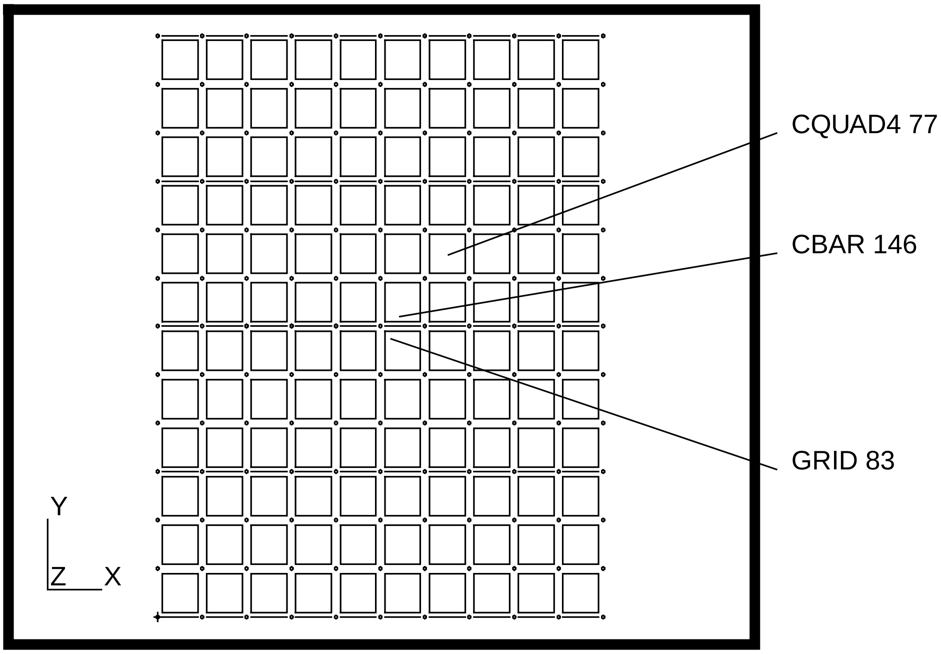
    nastran plate.bdf scr=yes

    Monitor the run using the UNIX **`ps`** command.

15a. When the run is completed, edit the **`plate.f06`** file and search for the word **FATAL**. If none exists, search for the word **WARNING**. Determine whether or not existing WARNING messages indicate modeling errors.

15b. While still editing **`plate.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 83 (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 146?

axial stress = ____________

Search for the word:

Q U A D (spaces are necessary)

What are the centroidal Von Mises stresses for CQUAD4 77?

-(thk/2): stress = ____________

+(thk/2): stress = ____________

16. Proceed with the Reverse Translation process, that is, importing the **`plate.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...

Filter

Selected Results File: *select the desired* ***.op2*** *file*

OK

Apply

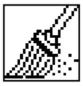
When translation is complete the Heartbeat turns green. Bring up the *Results* form.

◆ **Results**

Action:	**Create**
Object:	**Quick Plot**

Choose the desired result case in the *Select Result Cases* list and select the result(s) in the *Select Fringe Result* list and/or in the *Select Deformation Result* list. And hit **Apply** to view the result(s) in the viewport.

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 broom icon.



Reset Graphics

Quit MSC/PATRAN when you have completed this exercise.