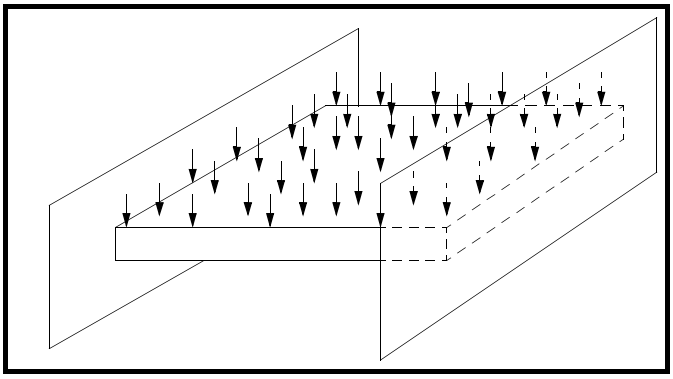
WORKSHOP 14c

Plate Characteristics



Objectives:

- Create a plate model.
- Manually define material and element properties.
- Fix the two ends of the surface.
- Apply pressure onto the top surface of the plate.
- Run the analysis using MSC.Nastran.

Model Description:

Because the loading from the previous two models was a bending load, there was no appreciable difference in modeling the plate with bending panels or shells. Apply a 1.0 inch enforced displacement in the X-Direction on the right edge of the plate.

The plate dimensions are given in Figure 14c.1 The loads and boundary conditions are shown in Figure 14c.2.

Figure 14c.1 - Geometry and Finite Elements

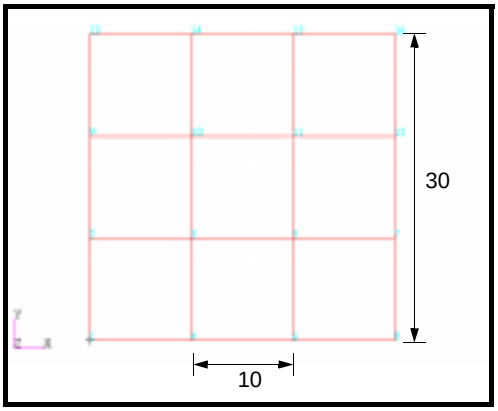


Figure 14c.2 - Constraints and Applied Forces

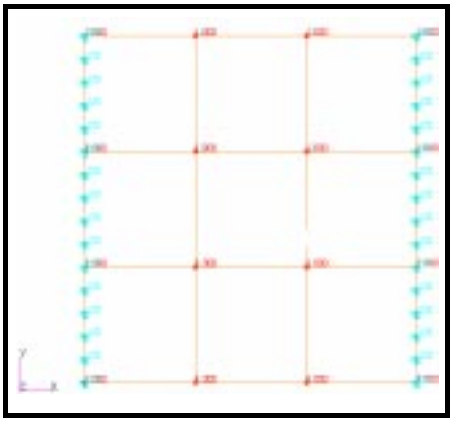


Table 14c.1 - Plate Properties

Plate Thickness	**900 square inch**
Elastic Modulus	**10E6 psi**
Poisson's Ratio	**0.3**

Suggested Exercise Steps:

- Open the model created in the previous exercise.
- Delete the pressure load on the
- Use the load and boundary condition sets to define a load case (SUBCASE).
- Prepare the model for a linear static analysis (SOL 101).
- Submit it for a linear static analysis.
- Review results.

Exercise Procedure:

1. Open the database **under_shear.db**

   File/Open...

   Existing Database Name: **under_Shear**

   OK

2. Delete the pressure load from the previous model.

   ◆ **Loads/BCs**

   Action: **Delete**

   Object: **Pressure**

   Sets to be Deleted Select the existing sets **P1**

   Apply

3. After deleted pressure, modify Displacements by following these steps in the **Load/BCs** form

   ◆ **Loads/BCs**

   Action: **Modify**

   Object: **Displacement**

   Type: **Nodal**

   Select Set to Modify

   pin

Change to a front view by selecting the **Front View** button on the toolbar

Front View

Modify Application Region

Geometry Filter: ◆ **FEM**

Select Nodes **Select the left edge of plate**

Application Region **Highlight all the nodes**

Hit **Space bar** on the keyboard

Add

OK

Apply

4. Create Displacement on the right edge of the plate

   ◆ **Loads/BCs**

   Action: **Create**

   Object: **Displacement**

   Type: **Nodal**

   New Set Name: **endforce**

   Input Data...

   Translations <T1,T2,T3> **<1.0, 0.0, 0.0>**

   OK

   Select Application Region...

   Geometry Filter: ◆ **FEM**

Select Nodes: **Select the right edge of plate**

Add

OK

Apply

5. 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: **under_shear**

Apply

An MSC.Nastran input file called bend_panel.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.

6. Submit the input file to MSC.Nastran for analysis

To submit the MSC.Nastran.bdf file for analysis, find an available UNIX shell window. At the command prompt enter: **nastran under_shear.bdf scr=yes.** Monitor the run using the UNIX **ps** command.When the run is completed, proceed with the Reverse Translation process, that is, importing the **under_shear.op2** results file into MSC.Patran. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

Action: **Read Output 2**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Available Files: **under_shear.op2**

OK

Apply

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

◆ **Results**

Action: **Create**

Object: **Quick Plot**

Now click on the Select **Results** icon.

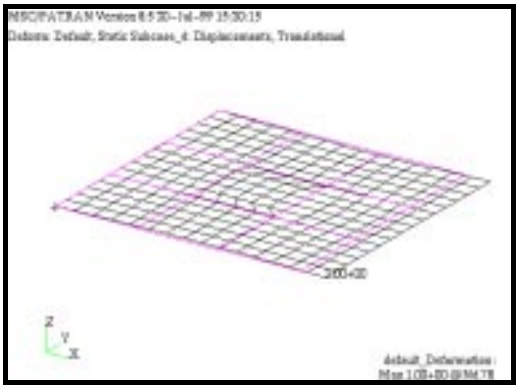
Select Results

Select Deformation Result: **Displacements, Translational**

Apply

The result should look like the following:

Figure 14c.1



1. Reset the graphics using the Reset Graphics tool.:

Reset Graphics

Quit MSC.Patran when finishing with this exercise.