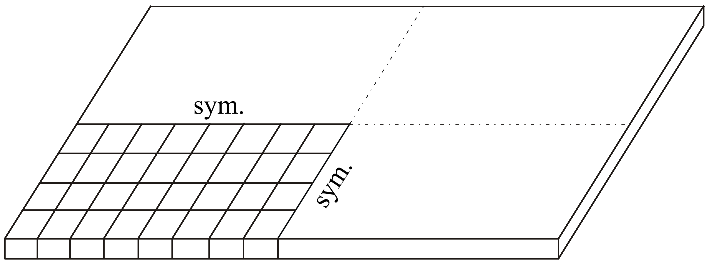
Workshop 4

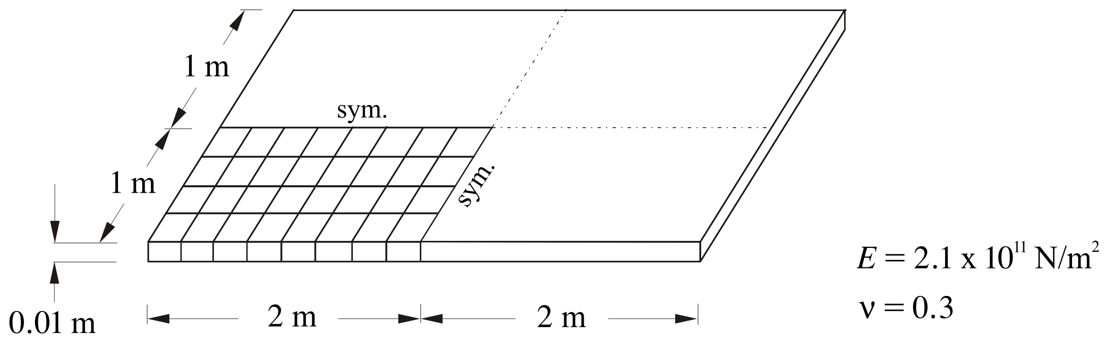
Linear Static Analysis of a Simply-Supported Plate



Objectives

- Create a geometric representation of a plate.
- Use the geometry model to define an analysis model comprised of plate elements.
- Run an MSC/NASTRAN linear static analysis.
- View analysis results.

Model Description



Consider the simply supported rectangular plate above, of thickness 0.01 m and sides 4 m and 2 m, subjected to a uniformly distributed load of 100 N/m^2.

Exercise Procedure

1. Start up **MSC/NASTRAN for Windows 4.5** and begin to create a new model.

   Double click on the icon for the **MSC/NASTRAN for Windows V4.5**.

   On the *Open Model File* form, select **New Model**.

   Turn off the workplane:

   Tools / Workplane (or **F2**) / ☐ **Draw Workplane / Done**

   View / Regenerate (or **Ctrl G**).

2. Create a material called **mat_1**.

   From the pulldown menu, select **Model / Material**.

Title	**mat_1**
Young's Modulus	**2.1e11**
Poisson's Ratio	**0.3**

   Select **OK / Cancel**.

   NOTE: In the *Messages Window* at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

3. Create a property called **prop_1** to apply to the members of the plate.

   From the pulldown menu, select **Model / Property**.

Title	**prop_1**
Material	**mat_1**

   Note that the default element type is **Plate** element, **not parabolic**.

Thickness, Tavg or T_1	**0.01**

   Select **OK / Cancel**.

4. Create the MSC/NASTRAN model for the plate (8×4 mesh of QUAD4).

   From the pulldown menu, select **Mesh / Between** (or **Ctrl B**).

Property	**prop_1**
Mesh Size / #Nodes / Dir. 1	**9**
Mesh Size / #Nodes / Dir. 2	**5**

   Select **OK**.

   Note that **Quad** is the default element shape. So, **Plate** + **not parabolic** (linear) + **Quad** = **QUAD4**. Due to symmetry considerations, just one quarter of the plate will be modelled.

	X:	*Y*:	*Z*:
Corner 1	*0*	*0*	*0*

   Select **OK**.

   Repeat this process for the other 3 corners.

X:	*Y*:	*Z*:	
2	*0*	*0*	**OK**
2	*1*	*0*	**OK**
0	*1*	*0*	**OK**

   To fit the display onto the screen, select **View / Autoscale / Visible** (or **Ctrl A**).

5. Create the model constraints.

   Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

   Model / Constraint / Set

Title	**constraint_1**

Select **OK**.

Now, define the relevant constraint for the model.

Model / Constraint / Nodal / Pick^ / Box

Select all 5 nodes on the left edge / **OK**.

On the *DOF* box, select

☐ **TX**	☐ **TY**	☑ **TZ**
☑ **RX**	☐ **RY**	☐ **RZ**

Select **OK**.

Pick^/ Box

Select all 5 nodes on the right edge / **OK**.

On the *DOF* box, select **X Symmetry / OK**.

Pick^/ Box

Select all 9 nodes on the bottom edge / **OK**.

On the *DOF* box, select

☐ **TX**	☐ **TY**	☑ **TZ**
☐ **RX**	☑ **RY**	☐ **RZ**

Select **OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine.

Pick^/ Box

Select all 9 nodes on the top edge / **OK**.

On the *DOF* box, select **Y Symmetry / OK**.

The same warning message above appears on the screen. Again, select **No** and, then, **Cancel**.

6. Create the model loading.

Like the constraints, a load set must first be created before creating the appropriate model loading.

Model / Load / Set (or **Ctrl F2**)

Title	**load_1**

Select **OK**.

Now, define the 100 N/m^2 surface load.

Model / Load / Elemental / Select All / OK.

Highlight **Pressure**

Load	**Pressure**	**100**

Select **OK**.

Face	**2**

Select **OK / Cancel**.

In order to visualize the loading, you may want to rotate the model.

View / Rotate (or **F8**) **/ Dimetric / OK**.

7. Run the analysis.

File / Analyze

Analysis Type	**Static**
Loads	✓ **load_1**
Constraints	✓ **constraint_1**
	✓ **Run Analysis**

Select **OK**.

When asked if you wish to save the model, respond **Yes**.

Be sure to set the desirable working directory.

File Name	**work_4**

Select **Save**.

When the MSC/ NASTRAN manager is through running, MSC/ NASTRAN for Windows will be restored on your screen, and the *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time. Then, select **Continue**.

8. Display the deformed plot on the screen.

You may now display the deformed plot. First, however, you may want to remove the load and boundary constraint markers.

View / Options / Quick Options (or **Ctrl Q**)

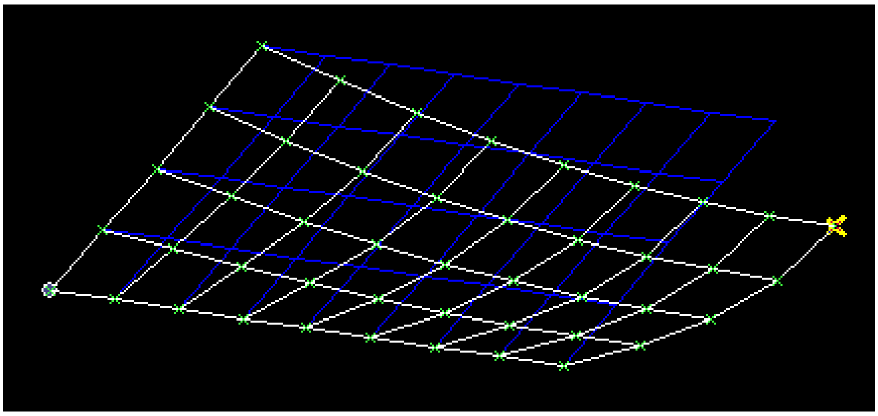
☐ **Pressure** / ☐ **Constraint / Done / OK**

Plot the deformation of the plate.

View / Select (or **F5**)

Deformed Style	**Deform**
	Deformed and Contour Data
Output Vectors / Deformation	**Total Translation**

Select **OK / OK**.



What is the center deflection?

The answer is easily given clicking **Off**, at the right bottom side of the screen, and then selecting **Node**.

The required deflection will come out when the cursor is left next to the center node.

9. Display the plate *X bending moment contour* on the screen.

View / Select (or **F5**)

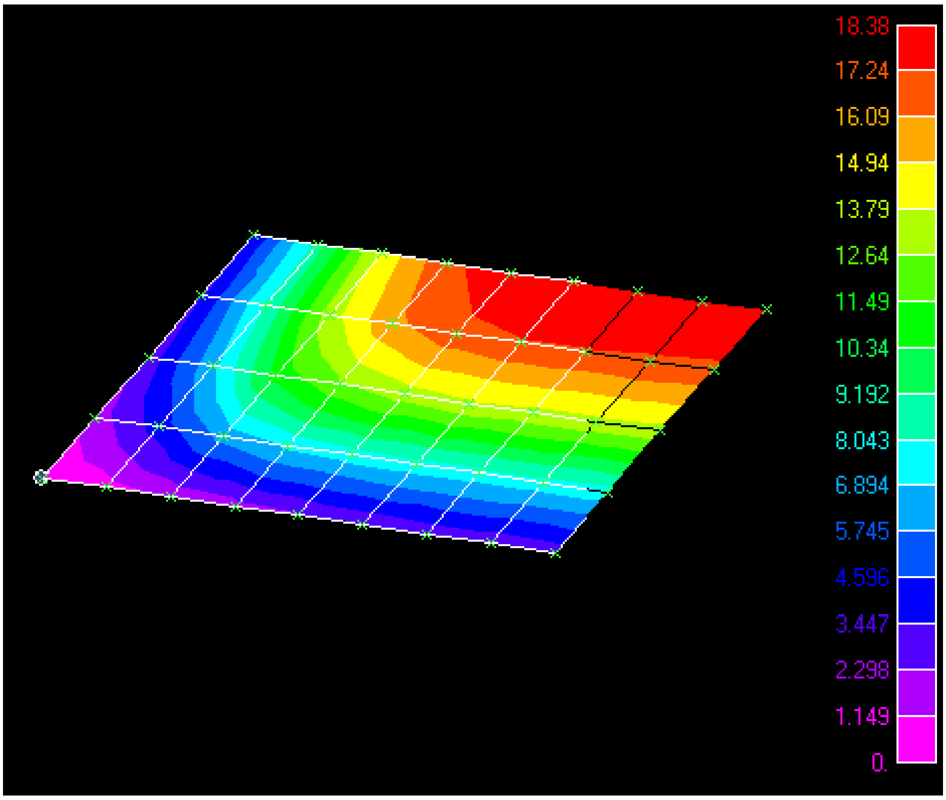
Deformed Style **None - Model Only**

Contour Style **Contour**

Deformed and Contour Data

Output Vectors / Contour **7211 Plate X Bending Moment**

Select **OK** / **OK**.



What is the X bending moment at the center of the plate?

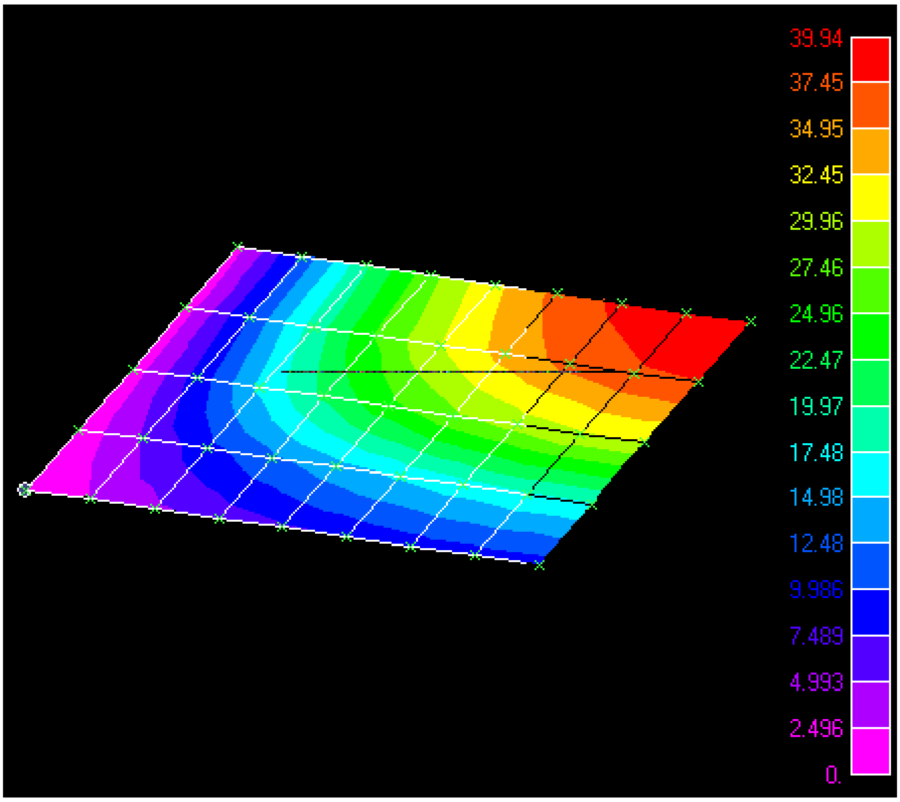
If you want, the procedure used to find the center deflection could be also applied here to answer this question.

10. Display the plate *Y bending moment contour* on the screen.

View / **Select** (or **F5**)

	Deformed and Contour Data
Output Vectors / Contour	**7212 Plate Y Bending Moment**

Select **OK** / **OK**.



What is the Y bending moment at the center of the plate?

The same procedure used to find the X bending moment could be also applied here to answer this question.

Return the model to the original display.

View / Select (or **F5**)

Deformed Style	**None - Model Only**
Contour Style	**None - Model Only**

Select **OK**

This concludes the exercise.

File / Save

File / Exit.

Answer

	MSC/NASTRAN	**Theoretical values**
w	-8.425×10^{-4}	-8.427×10^{-4}
M_x	18.26228	18.540
M_y	39.94327	40.673

Would you like to improve the result by refining the mesh?

NOTE: The theoretical values are the Navier solutions of the Kirchhoff plate theory for m, $n = 1, 3, \ldots, 101$ (EDI-32).