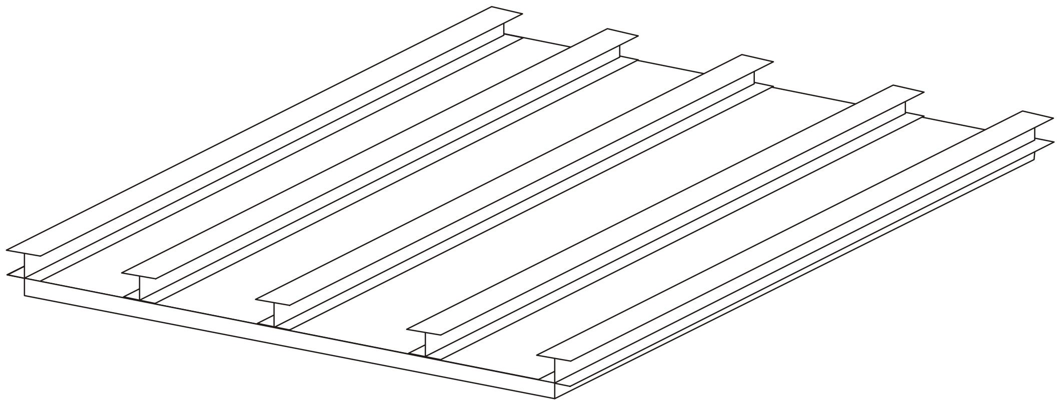
# Workshop 5

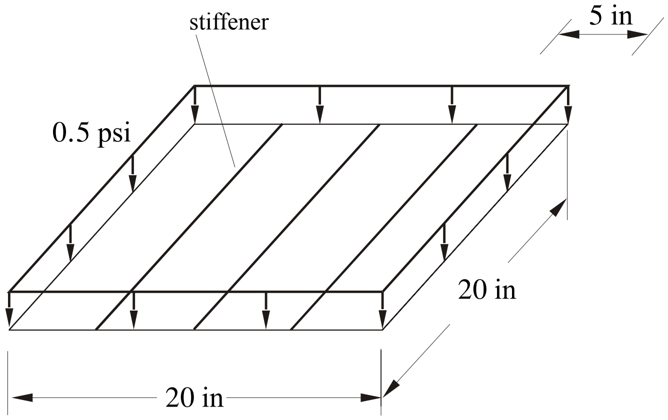
# Linear Static Analysis of a Simply-Supported Stiffened Plate

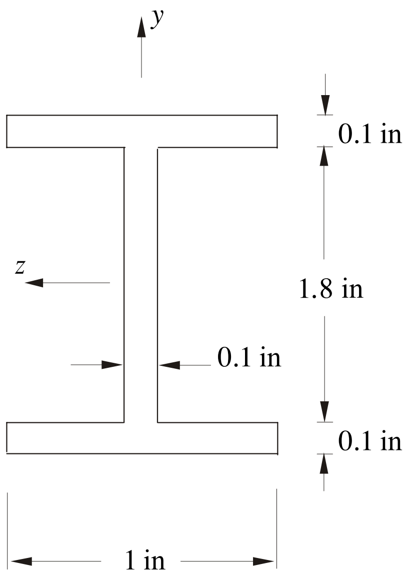


## Objectives

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

# Model Description



stiffener cross section

Consider the square stiffened plate above of 20 in side and 0.1 in thickness, with I-beam stiffeners mounted as shown. The structure is simply supported on its four corners and subjected to a uniformly distributed load of 0.5 psi 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 can account for this when you define the element properties for the stiffeners.

| | |
|---|---|
| Young's Modulus | $10.3 \times 10^6$ psi |
| Poisson's Ratio | 0.3 |
| Plate Thickness | 0.1 in |
| Bar Cross-Sectional Area | 0.38 in$^2$ |
| $I_z$ | 0.2293 in$^4$ |
| $I_y$ | 0.0168 in$^4$ |
| $J$ | 0.0013 in$^4$ |

# 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* | **10.3e6** |
   | *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 **plate** to apply to the members of the plate itself.

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

   | | |
   |---|---|
   | *Title* | **plate** |
   | *Material* | **mat_1** |

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

   | | |
   |---|---|
   | *Thickness, Tavg or* $T_1$ | **0.1** |

   Select **OK**.

4. Create a property called **stiffener** for the bar elements of the model.

   *Title* **stiffener**

   *Material* **mat_1**

   **Elem / Property Type**

   Change the property type from **Plate** element (default) to **Bar** element.

   *Line Elements* **Bar**

   Select **OK**.

   *Area* **0.38**

   $I_z$ **0.2293**

   $I_y$ **0.0168**

   $J$ **0.0013**

   Be certain that you understand the assumed bar orientation. Now, choose up to 4 points on cross-section, defined in element local yz plane, where stresses will be computed.

| Stress Recovery | | |
|---|---|---|
| Points | $y$ | $z$ |
| 1 | 1 | 0.5 |
| 2 | 1 | −0.5 |
| 3 | −1 | −0.5 |
| 4 | −1 | 0.5 |

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

5. Create the MSC/NASTRAN model for the plate (12 × 10 mesh of QUAD4).

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

   *Property* **plate**

   *Mesh Size / #Nodes / Dir. 1* **13**

| | |
|---|---|
| *Mesh Size / #Nodes / Dir. 2* | **11** |

Note that **Quad** is the default element shape. So, **Plate** + **not parabolic** (linear) + **Quad** = **QUAD4**.

Select **OK**.

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

Select **OK**.

Repeat this process for the other 3 corners.

| *X*: | *Y*: | *Z*: | |
|---|---|---|---|
| *20* | *0* | *0* | **OK** |
| *20* | *20* | *0* | **OK** |
| *0* | *20* | *0* | **OK** |

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

6. Create the MSC/NASTRAN model for the stiffeners (10 bar elements for each stiffener).

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

| | |
|---|---|
| *Property* | **stiffener** |
| *Mesh Size / #Nodes / Dir. 1* | **11** |
| *Corner Nodes* | **1** |
| | **131** |

Select **OK**.

When selecting the corner nodes, you may (if you wish) manually type their numbers. However, it is easier to use the graphic interface and select the nodes on the screen using the mouse. Click in the first *Corner Nodes* box and then select the nodes on the screen. Notice that the node nearest to the cursor is highlighted by a large yellow X - you don't have to click precisely on the node!

Now, specify the orientation vector for the bar elements.

| | $X$: | $Y$: | $Z$: |
|---|---|---|---|
| *Base* | 0 | 0 | 0 |
| *Tip* | 0 | 0 | 1 |

Select **OK**.

NOTE: In MSC/NASTRAN, the way to construct the element coordinate system is by defining an *orientation vector*, as explained in Workshop 2. The element lies on the local $x$ axis and the moments of inertia $I_1$ and $I_2$ are related to the bending about the local $z$ and $y$ axes, respectively.

Create the rest of the bar elements:

**Mesh / Copy / Element / Method / Type**

Select *L Bar* from the pulldown menu / **OK**

| | |
|---|---|
| *Repetitions* | 4 |

Select **OK**

| | $X$: | $Y$: | $Z$: |
|---|---|---|---|
| *Base* | 0 | 0 | 0 |
| *Tip* | 5 | 0 | 0 |

Select **OK**.

7. Remove coincidents grids from the model.

   Additional grids were created while generating bar elements. To eliminate these coincident grids, do the following:

**Tools / Check / Coincident Nodes / Select All / OK**

When asked if you wish to specify an additional range of nodes to merge, respond **No**.

| | | |
|---|---|---|
| *Options* | ✓ | **Merge Coincident Entities** |

Select **OK**.

As the *Messages Window* states, 55 nodes have now been merged.

8. Offset the stiffeners to correctly represent the model.

To properly model the bar stiffness, we will need to incorporate the bar offset property.

In order to examine the offsets, you may want to rotate the model.

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

**Modify / Update Elements / Offsets / Method / Type**

Select *L Bar* from the pulldown menu / **OK**

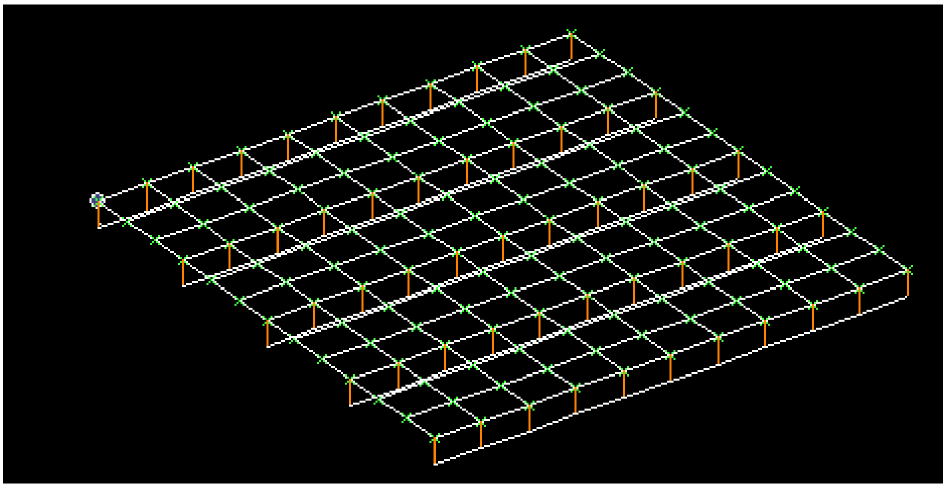
| | |
|---|---|
| ✓ | **Update End A** |
| ✓ | **Set End B = End A** |

Select **OK**

| | $X$: | $Y$: | $Z$: |
|---|---|---|---|
| *Base* | *0* | *0* | *0* |
| *Tip* | *0* | *0* | *-1.05* |

Select **OK**.

NOTE: The offset values is 1/2 the height of the cross section and 1/2 the thickness of the plate.



9. 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**

   Select the four extreme corners, **Nodes 144, 13, 154 and 198 / OK**.

   On the *DOF* box, select all translations.

   ✓ **TX** ✓ **TY** ✓ **TZ**

   Select **OK / Cancel**.

   Notice that the constraint appears on the screen at Nodes 144, 13, 154 and 198, fixing the 1, 2 and 3 directions (corresponding to TX, TY and TZ).

10. 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 0.5 psi surface load.

**Model / Load / Elemental / Select All / OK**.

Highlight **Pressure**

*Load* **Pressure** | **0.5**

Select **OK**.

*Face* | **2**

Select **OK / Cancel**.

11. 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_5**

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 detail this time. Then select **Continue**.

12. List the results of the analysis.

To list the results, select the following:

**List / Output / Unformatted / Select All / OK**

Unselect **All Vectors** and instead select **T3 Translation**

☐ **All Vectors / T3 Translation / OK**

NOTE: You may want to expand the message box in order to view the results.

Answer the following questions using the results. The answers are listed at the end of the exercise.

What is the minimum displacement (maximum absolute value) in the Z direction and where it occurs?

Min. Disp. Z = ____________________ Nodes = ____________________

Also, repeat the **List / Output / Unformatted** procedure above to find the answer to the second question.

What is the maximum von Mises stress on the top of the plate?

Max. von Mises Stress = ____________________

13. Display the deformed plot on the screen.

Finally, 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**)

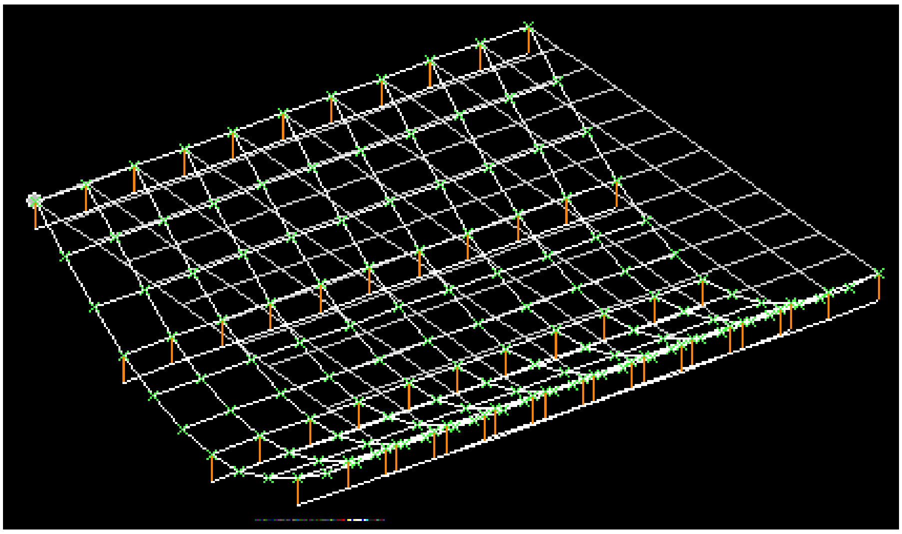
☐ **Pressure** / ☐ **Constraint / Done / OK**

Plot the deformation of the structure.

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

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

Select **OK / OK**.

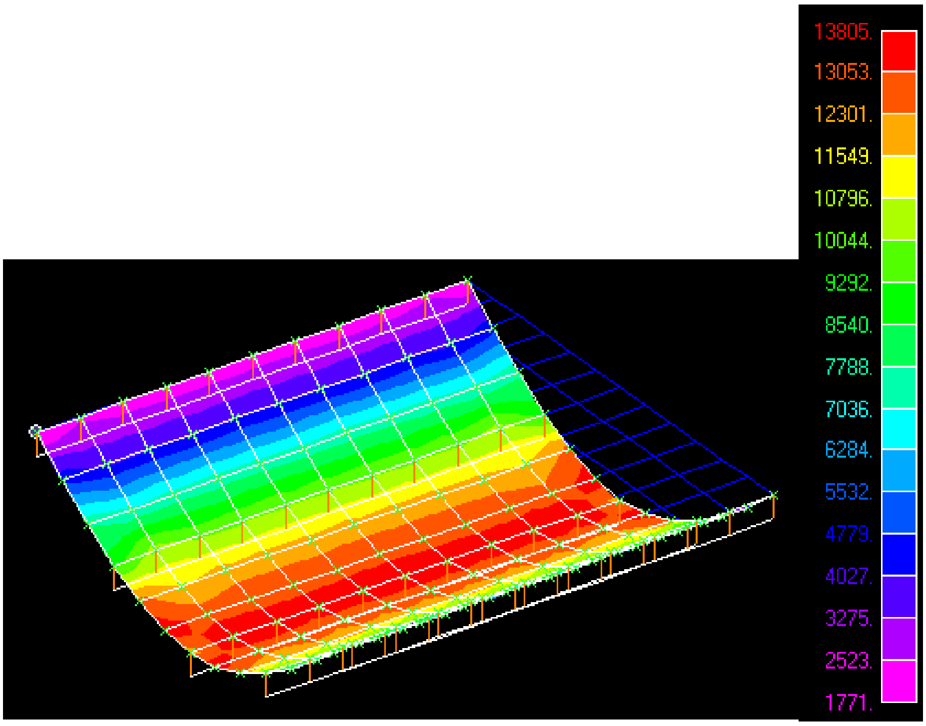


14. Add the contour plot of von Mises stress to the deformed plot.

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

| | |
|---|---|
| *Contour Style* | **Contour** |
| | **Deformed and Contour Data** |
| *Output Vectors / Contour* | **7033 Plate Top von Mises Stress** |

Select **OK / OK**.




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

| | |
|---|---|
| **Min. Disp. Z** | -1.10626 at nodes 7 and 176 |
| **Max. von Mises Stress** | 13518.5 |

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

NOTE: The von Mises stress is defined as

$$\sigma_e = \sqrt{\frac{1}{2}\left[(\sigma_x - \sigma_y)^2 + (\sigma_x - \sigma_z)^2 + (\sigma_y - \sigma_z)^2\right] + 3\left(\tau_{xy}^2 + \tau_{xz}^2 + \tau_{yz}^2\right)}.$$

It is an invariant quantity with respect to rotation of the coordinate axes to which the stress is referred. Some failure theories state that yielding begins when $\sigma_e$ reaches a limiting value. Since it is common to neglect $\sigma_z$, $\tau_{xz}$ e $\tau_{yz}$ for plates,

$$\sigma_e = \sqrt{\sigma_x^2 - \sigma_x\sigma_y + \sigma_y^2 + 3\tau_{xy}^2}.$$