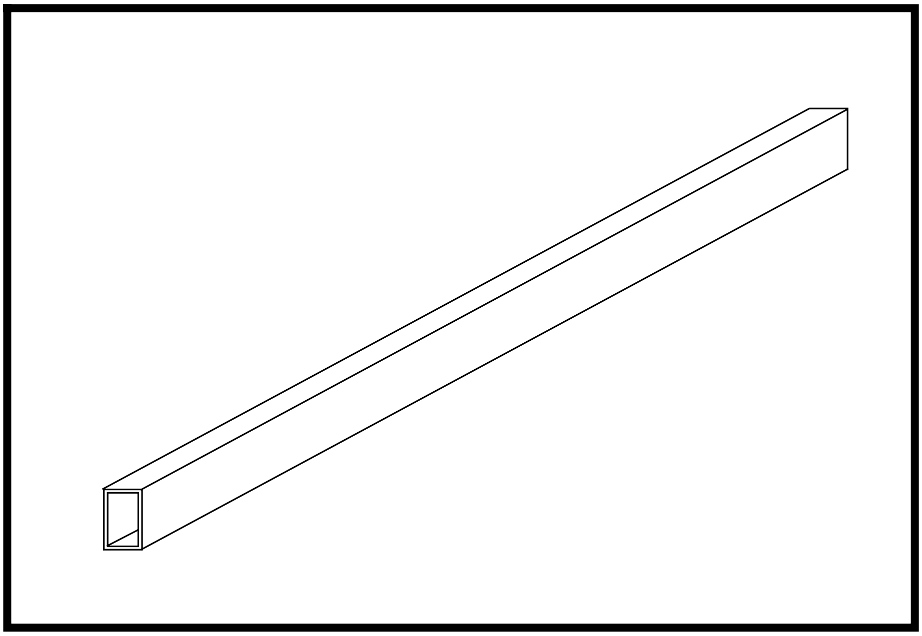
# WORKSHOP 6

# *Linear Static Analysis of a Cantilever Beam (SI Units)*



## Objectives:

- Create a geometrical representation of a cantilever beam.
- Use this geometry model to define an MSC/NASTRAN analysis model comprised of bar elements.
- Run an MSC/NASTRAN linear static analysis.
- Visualize analysis results.

# Model Description:

Below is a finite element representation of the beam structure shown on the title page. The beam has a hollow, rectangular cross-section as shown below in View A-A. The wall thickness is constant. The span of the beam is 5 m and has a fixed boundary constraint at X = 0 and a tip force of 1000 N is applied at X = 5 m in the negative Y-direction. The beam undergoes pure bending as a result of this applied load.

**Figure 6.1 -** Node Coordinates, LBCs and Element Connectivities

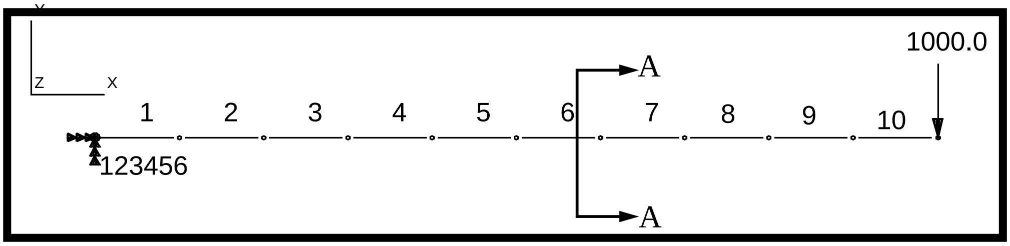



**Figure 6.2 -** Beam Cross Section

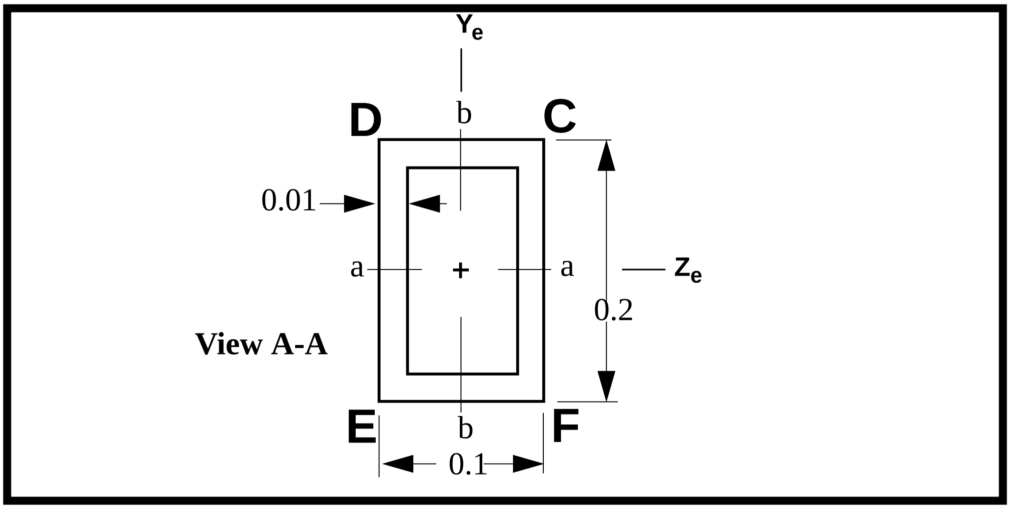



**Table 6.1** -Material Properties

| | |
|---|---|
| **Elastic Modulus:** | **7.1e10 N/m$^2$** |
| **Poisson Ratio:** | **0.3** |
| **Density:** | **2.650e4 N/m$^3$** |
| **Area:** | **5.600e-3 m$^2$** |
| **$I_{aa}$:** | **2.780e-5 m$^4$** |
| **$I_{bb}$:** | **8.990e-6 m$^4$** |
| **J:** | **2.090e-5 m$^4$** |

# Exercise Procedure:

1. Start up MSC/NASTRAN for Windows V3.0 and begin to create a new model.

Double click on the icon labeled **MSC/NASTRAN for Windows V3.0**

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

| | |
|---|---|
| *Open Model File:* | **New Model** |

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

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

**Model/Material...**

| | |
|---|---|
| *Title:* | **mat_1** |
| *Youngs Modulus:* | **7.1e10** |
| *Poisson's Ratio:* | **0.3** |
| *Mass Density:* | **2.65e4** |

**OK**

**Cancel**

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

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

**Model/Property...**

| | |
|---|---|
| *Title:* | **prop_1** |

To select the material, click on the **list** icon next to the databox and select **mat_1**.

| | |
|---|---|
| *Material:* | **1..mat_1** |

**Elem/Property Type...**

Change the property type from plate elements (default) to bar elements.

*Line Elements:* ● **Bar**

**OK**

*A:* **5.6e-3**

*I1:* **2.78e-5**

*I2:* **8.99e-6**

*J:* **2.09e-5**

NOTE: Be certain that you understand the assumed bar orientation.

| *Stress Recovery:* | | |
|---|---|---|
| | **Y** | **Z** |
| **1** | **.1** | **.05** |
| **2** | **.1** | **-.05** |
| **3** | **-.1** | **-.05** |
| **4** | **-.1** | **.05** |

**OK**

**Cancel**

4. Create the necessary NASTRAN geometry.

**Mesh/Between...**

*Property:* **1..prop_1**

*Mesh Size/ #Nodes:* **11**

**OK**

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

**OK**

| | *X:* | *Y:* | *Z:* |
|---|---|---|---|
| *Corner 2:* | **5** | **0** | **0** |

**OK**

Now, specify the orientation vector for the bar elements.

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

**OK**

To fit the display onto the screen, use the **Autoscale** feature.

**View/Autoscale**

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**

**OK**

Now define the fixed end of the model.

**Model/Constraint/Nodal...**

Select **Node 1**.

**OK**

On the *DOF* box, select all 6 boxes.

☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

**OK**

**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...**

| | |
|---|---|
| *Title:* | **load_1** |

**OK**

Now define the relevant loading conditions.

**Model/Load/Nodal...**

Select **Node 11**.

**OK**

Highlight **Force**.

| | |
|---|---|
| | **Force** |
| *FY* ☒ | **-1000** |

**OK**

**Cancel**

7. Submit the model for analysis.

**File/Export/Analysis Model...**

| | |
|---|---|
| *Analysis Format/Type:* | **1..Static** |

**OK**

Be sure to set the directory to **C:\temp**.

*File Name:* **beam**

**Write**

☒ **Run Analysis**

**OK**

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

**Yes**

*File Name:* **beam**

**Save**

When the MSC/NASTRAN manager is through running, MSC/NASTRAN 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.

**Continue**

8. View the text results of the analysis.

To list the results, select the following:

**List/Output/Unformatted...**

**Select All**

**OK**

Unselect **All Vectors** and instead select **T2 Translation**.

☐ **All Vectors, or**

**3..T2 Translation**

**OK**

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

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

What is the Y displacement at the tip (Node 11)?

Disp Y = _______

You can make another list to find the answer to the second question.

To list the results, select the following:

**List/Output/Unformatted...**

**Select All**

**OK**

Unselect **All Vectors** and instead select **Bar EndA Max Comb Stress**.

☐ **All Vectors, or**

**3109..Bar EndA Max Comb Stress**

**OK**

What is the maximum positive bending stress?

Max Pos Bending Stress = _______

9. Display the deformed plot on the screen.

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

**View/Options...**

**Quick Options...**

**Labels Off**

☐ **Load - Force**

☐ **Constraint**

**Done**

**OK**

Plot the deformation of the beam.

**View/Select...**

*Deformed Style:* ● **Deform**

**Deformed and Contour Data...**

*Output Vectors/Deformation* **3..T2 Translation**

**OK**

**OK**

10. Theory.

## Tip Deflection - Y Component:

Tip defl. $Y = -PL^3/3EI$ where:

| | | | |
|---|---|---|---|
| P | = tip load | = 1000 N |
| L | = beam length | = 5 m |
| E | = Elastic Modulus | = 7.1e+10 $N/m^2$ |
| I | = Moment of Inertia | = 2.78e-5 $m^4$ |

Tip defl. $Y = -[(1000)(5)^3]/[3(7.1e+10)(2.78e-5)]$ m

= -2.1110e-2 m

## Maximum Positive Bending Stress:

Max stress $\sigma_{max}$= abs[PLc/I] where:

| | | |
|---|---|---|
| P | = tip load | = 1000 N |
| L | = beam length | = 5 m |
| c | = extreme fiber distance | = .1 m |
| I | = Moment of Inertia | = 2.78e-5 $m^4$ |

Max stress $\sigma_{max}$= abs[(1000)(5)(.1)/(2.7e-5)] $N/m^2$

= 1.7986e+7 $N/m^2$

| | |
|---|---|
| **Y Disp** | -0.0211 |
| **Max Stress** | 1.7986 e+7 |