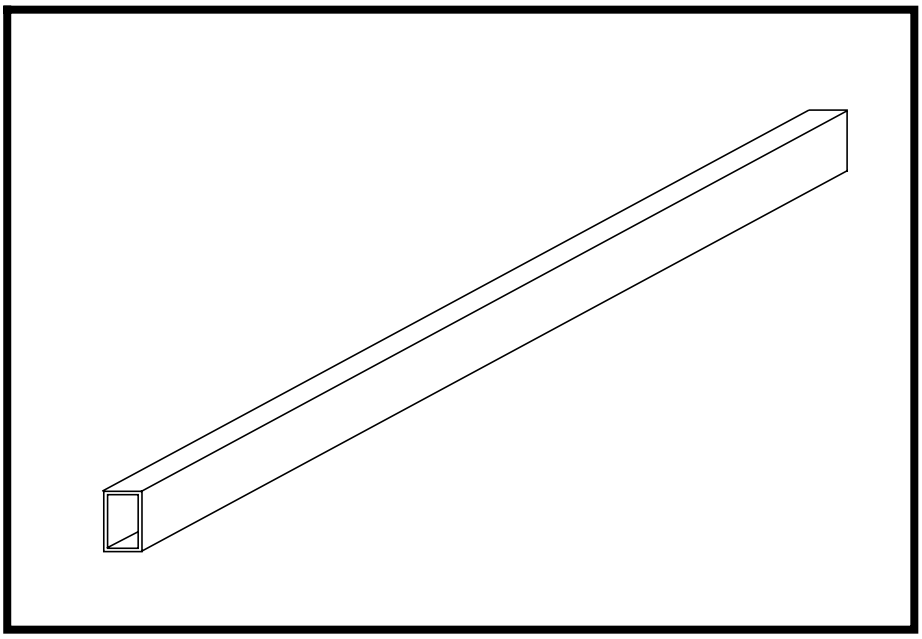
WORKSHOP 10

Cantilever Beam With Subcases



Objectives:

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

Model Description:

Below is a finite element representation of the beam structure shown on the title page. The beam hasan I-shaped cross-section as shown below in View A-A. The thickness is constant. The beam undergoes pure bending as a result of the three applied load cases.

Figure 10.1 - Node Coordinates and Element Connectivities

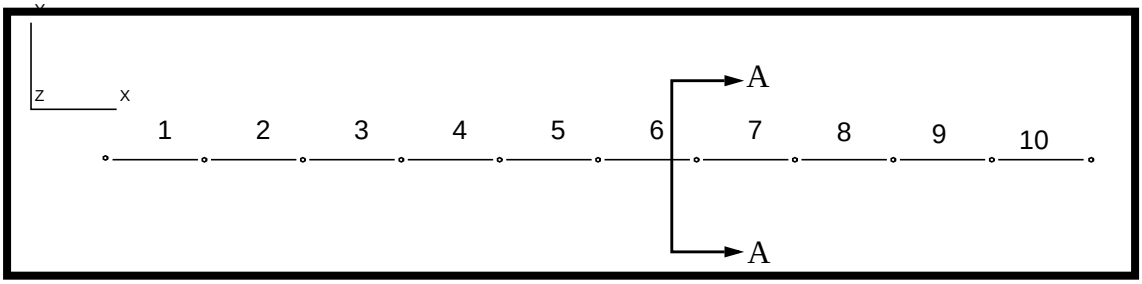


Figure 10.2 - Beam Cross Section

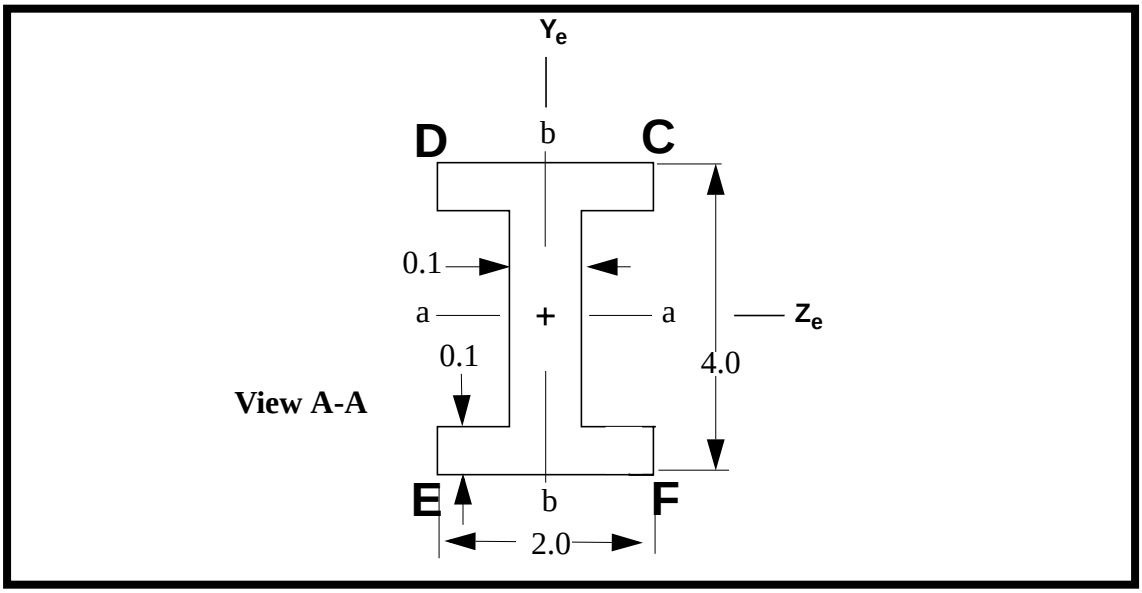


Table 10.1 -Material Properties

Elastic Modulus:	**30e6**
Poisson Ratio:	**0.3**
Hight of Beam:	**4**
Width of Beam:	**2**
Thickness:	**0.1**

Table 10.2 -Load Cases

Loading and Boundary Conditions (LBC)

1. Fixed at right, free at left.

2 Simply supported at both ends (be careful with the rigid body motion in RX, it is fatal error again in NASTRAN V70).

3 One nodal force at left.

4 One nodal force at center.

Load Cases

- Case 1 LBC 1 & 3
- Case 2 LBC 2 & 4
- Case 3 LBC 1, 3, & 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: **30e6**

Poisson's Ratio: **0.3**

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 ● **Beam**

OK

Shape...

Shape: **I - Beam or Wide Flange(W) Section**

Height, H: **4**

Width: **2**

Width, Bottom: **2**

Thickness, Top: **0.1**

Thickness, Bottom: **0.1**

Thickness: **0.1**

Orientation Direction(Y) ● **Right**

OK

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:	**100**	**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 Load Case 1.

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 11 (right end)**.

OK

Fixed

OK

Cancel

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 1 (left end)**.

OK

Highlight **Force**.

Force

FY ☒ **-2**

OK

Cancel

6. Create Load Case 2.

Constraint set 2 needs to be created. Do so by performing the following:

Model/Constraint/Set...

ID: **2**

Title: **constraint_2**

OK

Now define the fixed end of the model.

Model/Constraint/Nodal...

Select **Node 11 (right end) and Node 1 (left end)**.

OK

☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX**

OK

Cancel

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

Model/Load/Set...

ID: **2**

Title: **load_2**

OK

Now define the relevant loading conditions.

Model/Load/Nodal...

Select **Node 6 (center of beam)**.

OK

Highlight **Force**.

Force

FY ☒ **-1**

OK

Cancel

To clear the display and view the load set, use **View/Redraw**.

View/Redraw

7. Create Load Case 3.

Load set 3 must first be created before creating the appropriate model loading. .

Model/Load/Set...

ID: **3**

Title: **load_3**

OK

Since load set 3 is just a combination of load sets 1 and 2, the **load set combine** command can be utilized. .

Model/Load/Combine...

From/From Set: **1..load_1**

To/To Set: **3..load_3**

More

From/From Set: **2..load_2**

Last One

To clear the display and view the load set, use **View/Redraw**.

View/Redraw

8. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Type: **1..Static**

OK

Change the directory to **C:\temp**.

File name: **workshop10**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Beam with Subcases**

OK

Under *Analysis Case Requests*, enter the following:

SUBCASE ID:	**1**
☒ **Loads =**	**1..load_1**
☒ **Constraint =**	**1..constraint_1**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID:	**2**
☒ **Loads =**	**2..load_2**
☒ **Constraint =**	**2..constraint_2**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID:	**3**
☒ **Loads =**	**3..load_3**
☒ **Constraint =**	**1..constraint_1**

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

Done

OK

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

Yes

File name: **workshop10**

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

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 Set: **#..MSC/NASTRAN Case #**

Output Vectors/Deformation: **1..Total Translation**

OK

OK