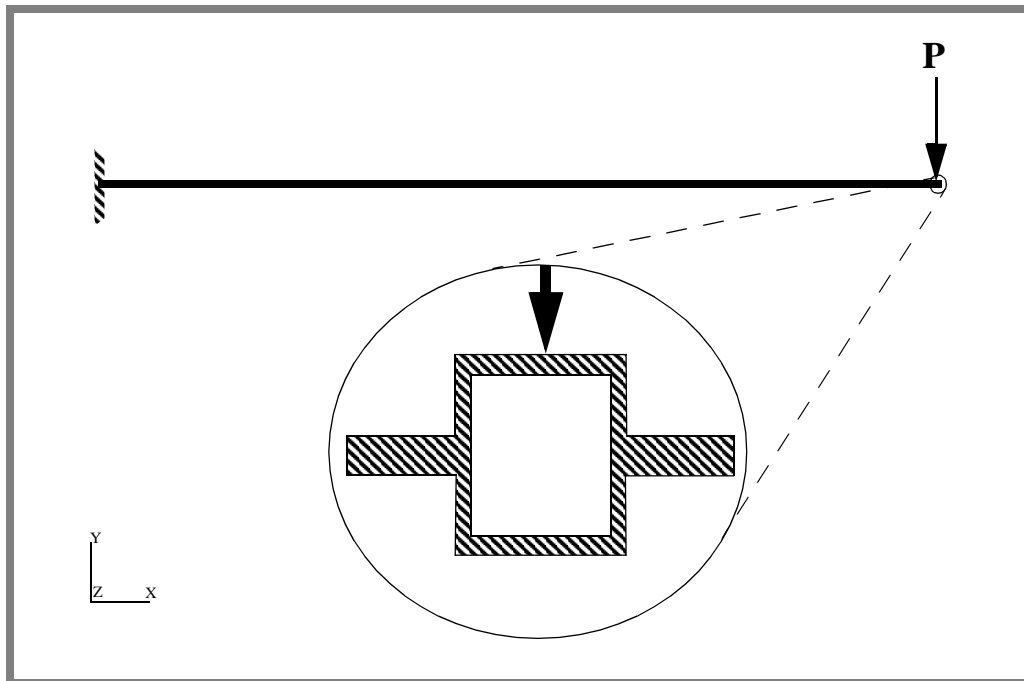

WORKSHOP 27

General Section Capability w/ Group & Layer (V4.0)



Objectives:

- Demonstrate the use of beam analysis using an arbitrary beam cross section.
- Run an MSC.Nastran for Windows 4.0 linear static analysis.
- Create an accurate deformation plot of the model.
- Analyze the results.

Model Description:

The figure below shows an arbitrary beam cross-section. The section properties will be assigned to a simple line model of a cantilever beam. The beam will be fixed at one end while a total load of 100 lbs is applied to the other end. Figure 27.1 displays the schematics of the arbitrary beam cross-section. Table 27.1 below displays all the necessary properties of the model.

Figure 27.1 - Schematic of an Arbitrary Beam Cross Section

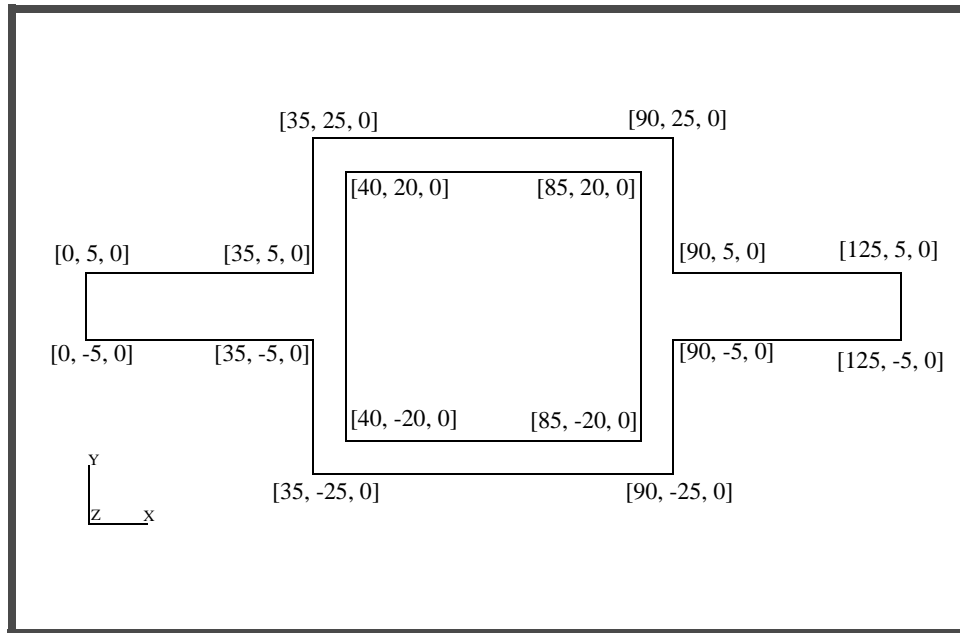


Table 27.1 - Cantilever Beam Properties

Total Length of Beam:	1000 inches
Elastic Modulus:	30E+06 psi
Poisson Ratio:	0.33
Total Force on End of Beam:	100 lb

Suggested Exercise Steps:

- Create the model using appropriate geometry.
- Apply arbitrary beam properties to a cantilever beam.
- Prepare the model for a linear static analysis.
- Generate an input file and submit it to the MSC.Nastran
Generate a deformation plot.
- Review the results.

Exercise Procedure:

1. Start up MSC.Nastran for Windows 4.0 and create a new model.

Start the MSC.N4W by double-clicking on the MSC.N4W icon. When the *Open Model File* dialog box appears, choose **New Model**.

Open Model File:

New Model

We are going to create a new layer on which to place the cross-section geometry. Later in the exercise, we will create a different layer to model the actual beam model.

View/Layers...

● Show Visible Layers Only

New Layer...

ID:

Title:

OK

Active Layer:

Hidden Layers: (highlight)

Show ->

OK

2. Define a material for the model.

Model/Material...

Title:

Youngs Modulus, E:

Poisson's Ratio:

OK

Cancel

-
3. Create the cross-section geometry. Start with the outer curves of the cross-section.

Geometry/Point...

Enter the following 12 points.

X:	<input type="text" value="0"/>	Y:	<input type="text" value="5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="0"/>	Y:	<input type="text" value="-5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="35"/>	Y:	<input type="text" value="-5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="35"/>	Y:	<input type="text" value="5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="35"/>	Y:	<input type="text" value="-25"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="35"/>	Y:	<input type="text" value="25"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="90"/>	Y:	<input type="text" value="25"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="90"/>	Y:	<input type="text" value="-25"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="90"/>	Y:	<input type="text" value="-5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="90"/>	Y:	<input type="text" value="5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="125"/>	Y:	<input type="text" value="5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="125"/>	Y:	<input type="text" value="-5"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>

View/Autoscale <Ctrl+A>

Turn off the workplane. **Right click** on screen.

Uncheck **Draw Workplane**.

Draw Workplane

View/Autoscale <Ctrl+A>

- Now create curves using the previously created points to match the pattern given in the model description.

Geometry/Curve - Line/Points...

Hint: To speed up the creation of a series of lines, *single-click* one point and *double-click* another point. This procedure will automatically execute the command without you needing to press the OK button every time. Repeat this procedure until you have created the pattern shown in Figure 1.1.

When done creating lines:

Cancel

- Now create the inner curves of the cross-section.

Geometry/Point...

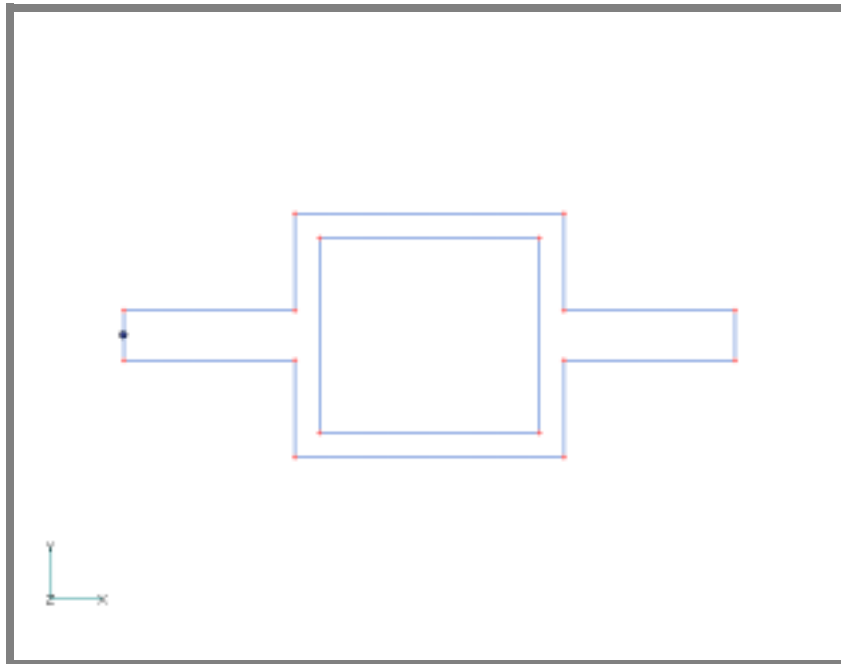
Enter the following four points.

X:	<input type="text" value="40"/>	Y:	<input type="text" value="20"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="40"/>	Y:	<input type="text" value="-20"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="85"/>	Y:	<input type="text" value="-20"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>
X:	<input type="text" value="85"/>	Y:	<input type="text" value="20"/>	Z:	<input type="text" value="0"/>	<input type="text" value="OK"/>

Cancel

Repeat the same procedure as in Step 4 in order to create the inner curves of the cross section.

Figure 27.2 - The completed curves of the cross-section.



6. Now create a surface from the boundary curves.

Geometry/Boundary Surface...

Select All

OK

Cancel

7. Now use this surface as a cross section for the beam properties.

Model/Property...

Elem/Property Type...

Line Elements:

Beam

OK

Title:

prop1

Material: (pull down)

1..mat1

Shape...

Shape: (pulldown)

Orientation Direction (y):

General Section...

Up

Surface...

Click on the cross-section surface.

OK

Define the section Y axis. This is just for the cross section orientation. We will define the beam orientation later in the exercise.

<i>Base:</i>	X:	<input type="text" value="0"/>	Y:	<input type="text" value="0"/>	Z:	<input type="text" value="0"/>
<i>Tip:</i>	X:	<input type="text" value="1"/>	Y:	<input type="text" value="0"/>	Z:	<input type="text" value="0"/>

OK

OK

OK

Cancel

- Now activate only the default layer in order to create and eventually mesh the beam. This will hide the cross section layer for clarity.

View/Layers...

<- Hide All

Hidden Layers: (highlight)

1..Default

Show ->

Active Layer: (pulldown)

1..Default

OK

-
9. Now we will create the actual cantilever beam that we are going to analyze.

Geometry/Curve - Line/ Coordinates...

X:	<input type="text" value="0"/>	Y:	<input type="text" value="0"/>	Z:	<input type="text" value="0"/>	<input type="button" value="OK"/>
X:	<input type="text" value="1000"/>	Y:	<input type="text" value="0"/>	Z:	<input type="text" value="0"/>	<input type="button" value="OK"/>

View/Autoscale <Ctrl+A>

10. Create finite elements for the beam.

Mesh/Mesh Control/Size Along Curve...

<input type="button" value="Select All"/>	
<input type="button" value="OK"/>	
<input checked="" type="radio"/> Element Size	<input type="text" value="100"/>
<input type="button" value="OK"/>	
<input type="button" value="Cancel"/>	

11. We are now ready to mesh the beam.

Mesh/Geometry/Curve...

Select the curve by clicking.

<i>Entity Selection:</i>	<input type="text" value="+17"/>
<input type="button" value="OK"/>	
<i>Property: (pull-down)</i>	<input type="text" value="1..prop1"/>
<input type="button" value="OK"/>	

Define the orientation of the cross section.

<i>Base:</i>	X:	<input type="text" value="0"/>	Y:	<input type="text" value="0"/>	Z:	<input type="text" value="0"/>
<i>Tip:</i>	X:	<input type="text" value="0"/>	Y:	<input type="text" value="1"/>	Z:	<input type="text" value="0"/>

OK

View/Options... <F6>

Options: (highlight)

Element Shape:

Element - Orientation/Shape

3..Show Cross Section

OK

12. Show the beam shaded in 3D.

View/Rotate... <F8>

Trimetric

OK

Click on the **View Style** toolbar button, then check **Render** and **Solid**.



View Style

Render

Solid

13. Now constrain the left end of the beam.

Model/Constraint/Nodal...

Title:

fixed

OK

Click the point on the *left* end of the beam.

OK

Fixed
OK
Cancel

14. Now create the load at the free end of the beam.

Model/Load/Nodal...

Title:

load

OK

Click the point on the *right* end of the beam.

OK

(highlight)

Force

FY:

-100

OK

Cancel

15. Run the analysis.

File/Export/Analysis Model...

Analysis Format/Type:

1..Static

OK

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

File Name:

general_section

Write

Additional Info:

Run Analysis

OK

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

Yes

File Name:

general_section

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 successfully, we will not bother with the details this time.

Continue

16. After the analysis, MSC.N4W automatically reads in the output results, which are available for a wide array of graphical and numerical post-processing.

View/Select... <F5>

Deformed Style:

Deform

Deformed and Contour Data...

Output Set: (pulldown)

1..MSC/NASTRAN Case 1

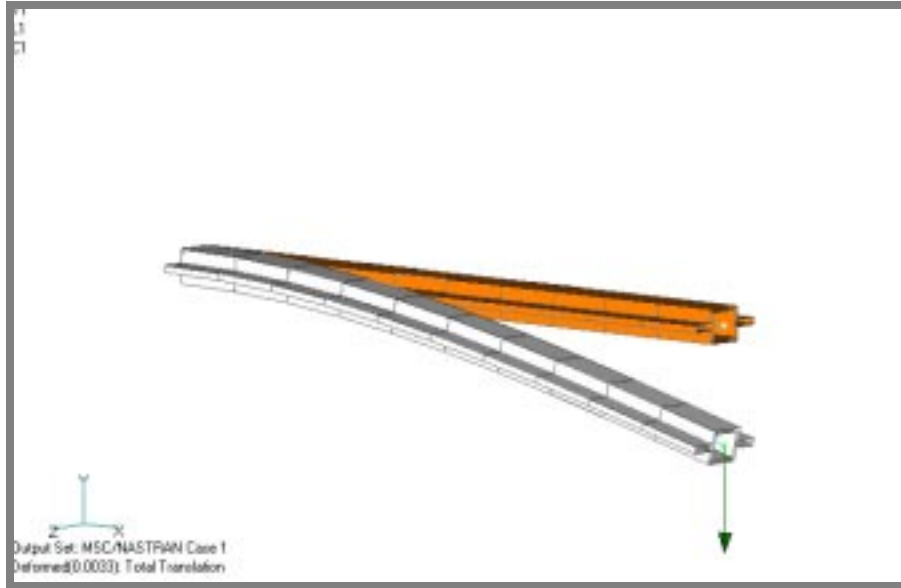
Deformation: (pulldown)

1..Total Translation

OK

OK

Figure 27.3 - Deformed Plot of the Beam



If you desire to compare the hand calculations to the Nastran results, the following will demonstrate their similarity. The equation used to determine the deflection of a cantilever beam under a load is:

$$\Delta\delta = \frac{Pl^3}{3EI}$$

Where **P** is the load, **l** is the length of the beam, **E** is the elastic modulus and **I** is the moment of inertia, where **I** = 338,750. We use this I in our case because when we selected the bar orientation vector <0 1 0> we are bending the beam in plane 1 of the elements coordinate system. The hand calculation and the linear analysis result from MSC.Nastran are exactly the same.

This concludes this exercise.

File/Save

File/Exit