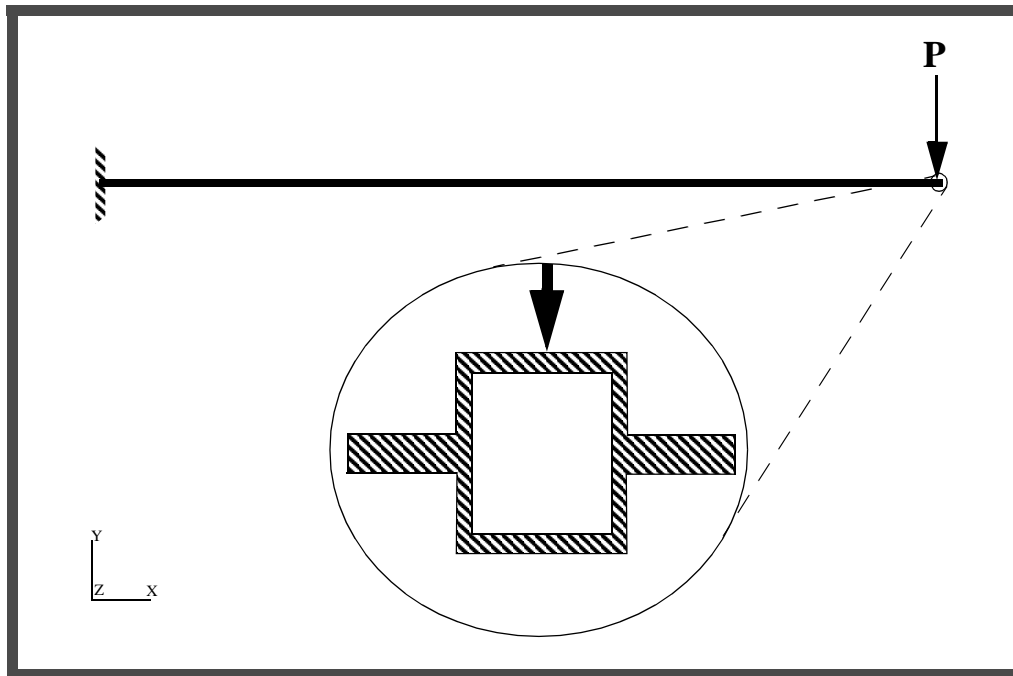

Lesson 18

Linear Static Analysis for a Beam of Arbitrary Cross-Section



Objectives:

- Demonstrate the use of beam analysis using an arbitrary beam cross section.
- Run an MSC/NASTRAN 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 given to a simple line model of a cantilever beam. The beam will be fixed at one end while a total load of 100 lbs will be applied at the other end. Figure 1.1 displays the schematics of the arbitrary beam cross-section. Table 1.1 below displays all the necessary properties of the model.

Figure 1.1 - Schematic of an Arbitrary Beam Cross Section

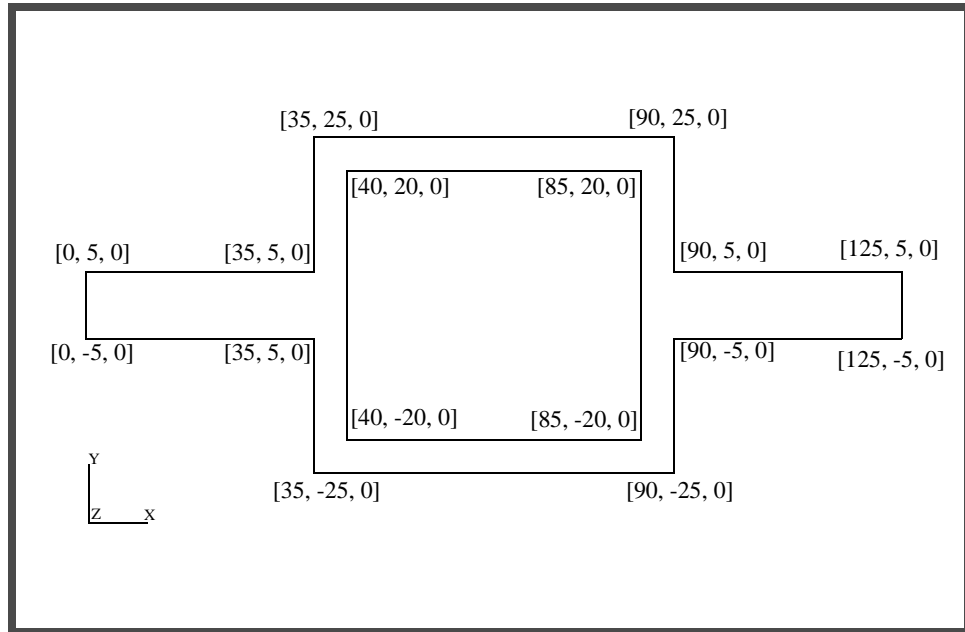


Table 1.1 -Cantilever Beam Properties

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 (SOL 101).
- Generate an input file and submit it to the MSC/NASTRAN solver for a linear static analysis.
- Generate a deformation.
- Review the results.

Exercise Procedure:

1. Create a new database called **prob1.db**.

File/New...

New Database Name

prob1

OK

In the **New Model Preference** form set the following:

Tolerance:

◆ **Default**

Analysis Code:

MSC/NASTRAN

Analysis Type:

Structural

OK

Whenever possible click **Auto Execute** (turn off).

Patran will calculate the properties for an arbitrary cross-section. The cross-section can be defined by a trimmed surface, and the trimmed surface can either be imported (through Iges for example) or it can be created directly in Patran. For this example we are going to actually create the trimmed surface that represents our cross-section.

2. Create the cross-section geometry.

Start with the outer curves of the cross-section first.

◆ **Geometry**

Action:

Create

Object:

Point

Method:

XYZ

Point Coordinates List

[0, 5, 0]

Apply

Point Coordinates List

[0, -5, 0]

Apply

To better view the points just created click on the following:



Point Size

Continue the same procedure for the remaining points of the geometry using the following table as a guide.

Table 1.2 - Point Coordinates List for Beam Cross-Section

Point ID	Point Coordinates	Point ID	Point Coordinates
3	[35, -5, 0]	10	[90, 25, 0]
4	[35, -25, 0]	11	[35, 25, 0]
5	[90, -25, 0]	12	[35, 5, 0]
6	[90, -5, 0]	13	[40, 20, 0]
7	[125, -5, 0]	14	[40, -20, 0]
8	[125, 5, 0]	15	[85, -20, 0]
9	[90, 5, 0]	16	[85, 20, 0]

You will need to click on the icon below to aid in creating the necessary curves.



Show Labels

Now create the curves using the previously created points.

◆ **Geometry**

Action:

Create

Object:

Curve

Method:

PWL

Point List

Point 1:12, 1

Apply

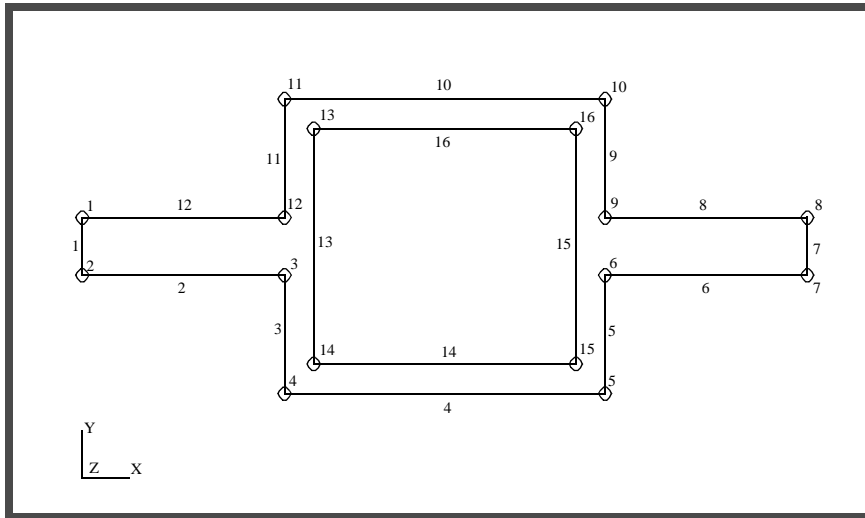
Point List

Point 13:16, 13

Apply

Your resulting cross section geometry on your screen should look similar to the following figure.

Figure 1.2 - Connectivities of Cross-Section



Now create the inner and outer curve loops of the cross-section by using the Chain method.

◆ **Geometry**

Action:

Create

Object:

Curve

Method:

Chain

■ **Delete Constituent Curves**

Curve List

Curve 1:12

Apply

(Select the outer curves of cross section.)

When a message box appears and asks if you wish to delete the original curves, click on **Yes**.

Yes

Curve List

Curve 13:16

Apply

(Select the inner curves of cross section.)

When a message box appears and asks if you wish to delete the original curves, click on **Yes**.

Yes

To create a surface out of the curves you will have to use the Trimmed method.

◆ **Geometry**

Action:

Create

Object:

Surface

Method:

Trimmed

Option:

Planar

■ **Use All Edge Vertices**

■ **Delete Outer Loop**

Outer Loop List

Curve 17

■ **Delete Inner Loops**

Inner Loop List

Curve 18

Apply

When a message box appears and asks if you wish to delete the original curves, click on **Yes** for each loop.

Yes

Yes

Turn off the labels and decrease the point size by clicking the following, respectively.



Hide Labels



Point Size

For clarity, create a group for the finished cross-section representation.

Group/Create...

Action:

Create

New Group Name

cross_section

■ **Make Current**

Group Contents:

Add Entity Selection

Entity Selection

Surface 1

Apply

(Select the cross section surface.)

Cancel

Now we will create the actual cantilever beam that we are going to analyze. The beam is going to be modeled by 1D elements, and to do this we are going to mesh a line entity.

◆ **Geometry**

Action:

Create

Object:

Curve

Method:

XYZ

Refer. Coordinate Frame

Coord 0

Vector Coordinates List

<1000, 0, 0>

Origin Coordinates List

[0, 50, 0]

3. Create the material property for the model.

◆ **Materials**

Action:

Create

Object:

Isotropic

Method:

Manual Input

Material Name

mat_1

Input Properties...

Elastic Modulus =

30E6

Poisson Ratio=

0.33

Apply

Cancel

4. Create the element property for the model by using the arbitrary beam cross-section.

◆ **Properties**

<i>Action:</i>	Create
<i>Dimension:</i>	1D
<i>Type:</i>	Beam
<i>Property Set Name</i>	prop_1

Input Properties...

■ **Associate Beam Section**

When a message box appears and asks if you wish to continue with the “Associate Beam Section” toggle on, click on **Yes**.

Yes

To associate our arbitrary beam cross-section to our line curve you must first click on the following icon.



<i>Action:</i>	Create
<i>Object:</i>	Arbitrary Shape
<i>Method:</i>	Boundary Loops
<i>New Section Name</i>	section_1
<i>Option:</i>	Select Surface
<i>Select Surface</i>	Surface 1

OK

The surface you select is the surface that represents the actual cross-section (the trimmed surface created in Step 2).

The Point matrix in the Beam Library form should now display the coordinates of the arbitrary beam cross-section.

Clicking on the Calculate/Display toggle in the Beam Library form should result in the computer displaying the Section Properties of the arbitrary beam.

Calculate/Display

Alternatively, the results given in the Section Properties can also be obtained through hand calculations. If comparing the hand calculations to the computer results was desired, tables 1.4 and 1.5 display the results.

Figure 1.3 - Simplified Cross-Section

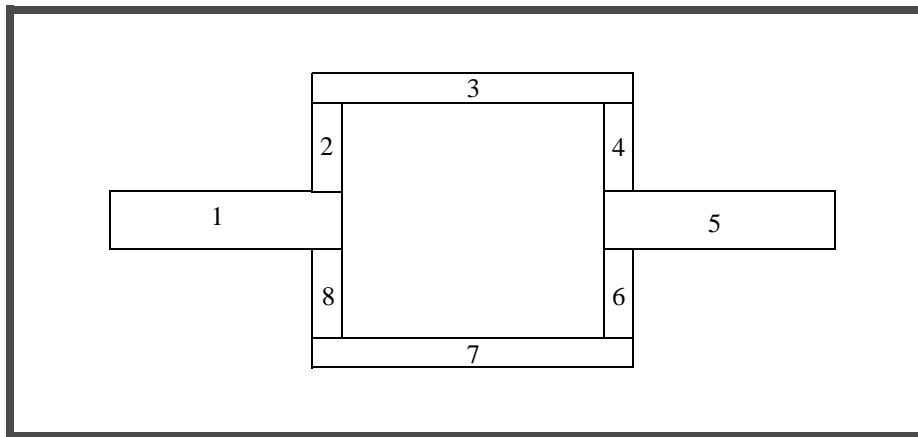


Table 1.4 - Hand Calculations for Arbitrary Beam

Section	Base	Height	Area	I_1	I_2	x_{cg}	y_{cg}	ax^2	ay^2
1	40	10	400	3333	53333	-43	0	722500	0
2	5	15	75	1406	156	-25	13	46875	11719
3	55	5	275	573	69323	0	23	0	139219
4	5	15	75	1406	156	25	13	46875	11719
5	40	10	400	3333	53333	43	0	722500	0
6	5	15	75	1406	156	25	-13	46875	11719
7	55	5	275	573	69323	0	-23	0	139219
8	5	15	75	1406	156	-25	-13	46875	11719
SUMS			1650	13438	245938			1632500	325313

The following equations were used to obtain the total moments of inertia,

$$I_n = \sum I_n + Ad^2$$

Using the above table and performing the above calculations gives the following total moments of inertias.

Table 1.5 - Total Moments of Inertia for Cross-Section

Total Section	
I₁	338750
I₂	1878438

Compare total I₁, I₂ and Area displayed in the Section Properties to what you get with hand calculations.

After viewing the new Section Properties finish with the Beam Library form by clicking on the following:

In the Input Properties form, enter the following:

Material Name

[Section Name]

Bar Orientation

Select Members

We are now done with the arbitrary beam cross-section. From this point on our attention will focus on the beam analysis. Since the arbitrary beam cross-section is no longer necessary, we will create a group for the cantilever beam and unpost the cross_section group.

First create a new group for the beam curve.

Group/Create...

Action:

Create

New Group Name

beam

Make Current

Unpost All Other Groups

Group Contents:

Add Entity Selection

Entity Selection

Curve 1

Apply

Cancel

5. Mesh the beam curve with five elements.

◆ Finite Elements

Object:

Create

Mesh

Type:

Curve

Global Edge Length

20

Element Topology

Bar 2

Curve List

Curve 1

Apply

You may now display the arbitrary beam in 3D by doing the following:

Display/Load/BC/Elem. Props...

Beam Display

3D: Full-Span

Apply

Click on the following icon to get a better perspective of the model.



Iso 1 View

Do the following to regain the 1D curve.

Beam Display

1D: Line

Apply

Cancel

Click on the following icon.



Front View

6. Now create the boundary conditions for the model.

Create the degree of freedom constraint for the model.

◆ **Loads/BCs**

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name

constraint_1

Input Data...

Translations < T1 T2 T3 >

< 0, 0, 0 >

Rotations < R1 R2 R3 >

< 0, 0, 0 >

OK

Select Application Region...

Geometry Filter

◆ **FEM**

Select Nodes

Node 1

Add

(Select end node on the left.)

OK

Apply

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

◆ **Loads/BCs**

Action:

Create

Object:

Force

Type:

Nodal

New Set Name

load_1

Input Data...

Force <F1 F2 F3>

<0, -100, 0>

OK

Select Application Region...

Geometry Filter

◆ **FEM**

Select Nodes

Node 6

Add

(Select end node on the right.)

OK

Apply

To clean up the display, use the following main menu icons:



Reset Graphics



Refresh Graphics

8. Generate an input file for the analysis.

Click on the **Analysis** radio button on the Top Menu Bar and set up the analysis as follows:

◆ **Analysis**

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name

probl

Solution Type...

Solution Type

◆ **LINEAR STATIC**

OK	
Subcase Create...	
<i>Available Subcases</i>	Default
Output Requests...	
<i>Form Type:</i>	Basic
<i>Output Requests</i>	STRESS(SORT1...
Delete	<i>(Select by highlighting.)</i>
<i>Output Requests</i>	SPCFORCES(SORT1...
Delete	<i>(Select by highlighting.)</i>
OK	
Apply	
Cancel	

Finally click apply from the **Analysis** form.

Apply

An input file called **prob1.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green.

Submit the input file for analysis:

9. Submit the input file to MSC/NASTRAN for an analysis.
 - 9a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob1.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.
 - 9b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob1.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.
10. When the analysis is completed, edit the **prob1a.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether the existing **WARNING** messages indicate any modeling errors.

Comparison of Results:

11. Compare the results obtained in the **.f06** file with the results on the following page:
12. **This ends the exercise for MSC/NASTRAN users. MSC/PATRAN Users should proceed to the next step.**
13. Proceed with the Reverse Translation process, that is, importing the **prob1.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ Analysis*Action:***Read Output2***Object:***Result Entities***Method:***Translate****Select Results File...***Selected Results File:***prob1.op2****OK****Apply**

14. Post process the results from the analysis.
Generate a deformation plot.

◆ Results*Action:***Create***Object:***Deformation**Now click on the **Select Results** icon.**Select Results***Select Result Case(s)***Default, Static Subcase***Select Deformation Result***Displacements, Translational**

Show As:

Component

■ YY

(Have YY the only toggle on.)

Now click on the **Plot Options** icon.



Plot Options

Coordinate Transformation:

None

Scale Factor

1.0

Apply

Your final deformation plot should resemble the following:

Figure 1.4 - Deformation Plot of Model



In order to see the deformation results accurately, set the Scale Interpretation to True Scale with a Scale Factor of 1.

Click on the **Display Attributes** icon.



Display Attributes

◆ True Scale

Scale Factor

1.0

■ *Show Undeformed*

Apply

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, for this case $I=I_1$ (Table 1.5). We use I_1 in our case because when we selected the bar orientation vector $\langle 0 \ 1 \ 0 \rangle$ we are bending the beam in plane 1 of the elements coordinate system.

Referring back to Figure 1.4, the hand calculation and the linear analysis result from Nastran are exactly the same.

Quit MSC/PATRAN when you have completed this exercise.

