



---

**APPENDIX B**



***PBEAML Exercise***



**Exercise Procedure:**

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

**File/New...***New Database Name:***pbeam****OK**

In the New Model Preferences form, set the following:

*Tolerance:*◆ **Default***Analysis Code:***MSC/NASTRAN***Analysis Type:***Structural****OK**

2. Create geometry for the finite element model.

◆ **Geometry***Action:***Create***Object:***Curve***Method:***XYZ***Vector Coordinate List:***< 50, 0, 0 >***Origin Coordinate List:***[ 0, 0, 0 ]****Apply**

3. Define nodes and elements by creating a mesh over the geometry.

◆ **Finite Elements***Action:***Create**

|                           |                |
|---------------------------|----------------|
| <i>Object:</i>            | <b>Mesh</b>    |
| <i>Type:</i>              | <b>Curve</b>   |
| <i>Global Edge Length</i> | <b>10</b>      |
| <i>Curve List</i>         | <b>Curve 1</b> |
| <b>Apply</b>              |                |

4. Create boundary conditions at Point 1.

For “cantilever” conditions at Point 1, constrain the translations and rotations in all directions:

◆ **Loads/BCs**

|                      |                     |
|----------------------|---------------------|
| <i>Action:</i>       | <b>Create</b>       |
| <i>Object:</i>       | <b>Displacement</b> |
| <i>Type:</i>         | <b>Nodal</b>        |
| <i>New Set Name:</i> | <b>cantilever</b>   |

**Input Data...**

|                                       |                      |
|---------------------------------------|----------------------|
| <i>Translations &lt;T1 T2 T3&gt;:</i> | <b>&lt;0 0 0&gt;</b> |
| <i>Rotations &lt;R1 R2 R3&gt;:</i>    | <b>&lt;0 0 0&gt;</b> |

**OK**

**Select Application Region...**

|                                   |                   |
|-----------------------------------|-------------------|
| <i>Geometry Filter:</i>           | <b>● Geometry</b> |
| <i>Select Geometric Entities:</i> | <b>Point 1</b>    |

**Add**

**OK**

**Apply**

## 5. Create load at Point 2 .

◆ **Loads/BCs***Action:***Create***Object:***Force***Type:***Nodal***New Set Name:***load****Input Data...***Force <F1 F2 F3>:***<0 -1000 0>****OK****Select Application Region...***Geometry Filter:*● **Geometry***Select Geometric Entities:***Point 2****Add****OK****Apply**

## 6. Define materials for the model.

◆ **Materials***Action:***Create***Object:***Isotropic***Method:***Manual Input***Material Name:***aluminum**

---

**Input Properties ...**

*Constitutive Model:*

**Linear Elastic**

*Elastic Modulus =*

**10e6**

*Poisson Ratio =*

**0.3**

**Apply**

**Cancel**

7. Create spatial field functions to describe *tapered dimensions* for I-beam.

◆ **Fields**

*Action:*

**Create**

*Object:*

**Spatial**

*Method:*

**PCL Function**

*Field Name:*

**height**

*Scalar Function:*

**3-0.02\*X**

**Apply**

*Field Name:*

**width**

*Scalar Function:*

**2-0.02\*X**

**Apply**

8. Associate “beam properties” to the finite element model.

◆ **Properties**

*Action:*

**Create**

*Dimension:*

**1D**

*Type:*

**Beam**

*Property Set Name:*

**taper**

*Option(s):***Tapered Section** **Input Properties ...***Material Name***m:aluminum***[Section Name]:***Dimensions**  (Value Type)**beam** (Value)*Bar Orientation:***Vector**  (Value Type)*(see Hint below)***Coord 0.2** (Value)

Our beam is aligned the x-direction. Entering “Coord 0.2” in *Bar Orientation* tells the cross section of the I-beam to align its web along the y-axis. The web can not align itself along the same axis that the beam is oriented. Thus, in our case, entering “Coord 0.1” would give an error in the Analysis.

**Hint:** In order to enter the Value (i.e. **Coord 0.2**) for *Bar Orientation*,  
(1) click:



(2) Then click the **magenta** coordinate marker located in the bottom left-hand corner of your viewport.

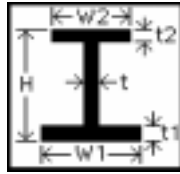
- Next, use **Beam Library** to create “I-beam” properties.  
Click:

*Action:***Create***Type:***Standard Shape***New Section Name:***beam**

- Choose “I-beam” cross section:



- Specify dimensions of the I-beam:



**Hint:** (1) Put cursor in the *H*, *W1*, and *W2* boxes, (2) then click “height” or “width” in the *Spatial Scalar Fields* box to enter the field dimensions.

|             |                 |
|-------------|-----------------|
| <i>H</i> :  | <b>f:height</b> |
| <i>W1</i> : | <b>f:width</b>  |
| <i>W2</i> : | <b>f:width</b>  |
| <i>t</i> :  | <b>0.1</b>      |
| <i>t1</i> : | <b>0.15</b>     |
| <i>t2</i> : | <b>0.15</b>     |

*Location Specification Options:*

**Select an Entity and a Parametric Location (C1) ▾**

- Display “section properties” of a desired cross section:

*C1 Parametric Location:*

(Vary the slider button position)

**.000**

**Calculate/Display**

**OK**

Ignore the warning message.

At the *Input Properties* form:

*Select Members:*

---

9. Submit model for linear static analysis.

Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown here:

◆ **Analysis**

*Action:*

**Analyze**

*Object:*

**Entire Model**

*Method:*

**Analysis Deck**

*Job Name:*

**pbeam**

**Apply**

An MSC/NASTRAN input file called **pbeam.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.

## Generating an input file for MSC/NASTRAN Users:

10. The generated input file (pbeam.bdf) should be similar to the following:

(Type “more pbeam.bdf” at UNIX shell window to compare.)

```

$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on February 24, 1998 at
$ 14:45:25.
ASSIGN OUTPUT2 = 'pbeam.op2', UNIT = 12
$ Direct Text Input for File Management Section
$ Linear Static Analysis, Database
SOL 101
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 23-Feb-98 at 19:15:57
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : Default
  SUBTITLE=Default
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
  SPCFORCES(SORT1,REAL)=ALL
  STRESS(SORT1,REAL,VONMISES,BILIN)=ALL
BEGIN BULK
PARAM POST -1
PARAM PATVER 3.
PARAM AUTOSPC YES
PARAM INREL 0
PARAM ALTRED NO
PARAM COUPMASS -1
PARAM K6ROT 0.
PARAM WTMASS 1.
PARAM,NOCOMPS,-1
PARAM PRTMAXIM YES
$ Direct Text Input for Bulk Data
$ Elements and Element Properties for region : taper
PBEAML 1 1 I
+ A
+ A 3. 2. 2. .1 .15 .15 YES
+ B
+ B 1. 2.8 1.8 1.8 .1 .15 .15
CBEAM 1 1 1 2 0. 1. 0.
PBEAML 2 1 I

```

|   |        |     |     |     |     |     |     |     |
|---|--------|-----|-----|-----|-----|-----|-----|-----|
| + | C      |     |     |     |     |     |     |     |
| + | C      | 2.8 | 1.8 | 1.8 | .1  | .15 | .15 | YES |
| + | D      |     |     |     |     |     |     |     |
| + | D      | 1.  | 2.6 | 1.6 | 1.6 | .1  | .15 | .15 |
|   | CBEAM  | 2   | 2   | 2   | 3   | 0.  | 1.  | 0.  |
|   | PBEAML | 3   | 1   |     | I   |     |     |     |
| + | E      |     |     |     |     |     |     |     |
| + | E      | 2.6 | 1.6 | 1.6 | .1  | .15 | .15 | YES |
| + | F      |     |     |     |     |     |     |     |
| + | F      | 1.  | 2.4 | 1.4 | 1.4 | .1  | .15 | .15 |
|   | CBEAM  | 3   | 3   | 3   | 4   | 0.  | 1.  | 0.  |
|   | PBEAML | 4   | 1   |     | I   |     |     |     |
| + | G      |     |     |     |     |     |     |     |
| + | G      | 2.4 | 1.4 | 1.4 | .1  | .15 | .15 | YES |
| + | H      |     |     |     |     |     |     |     |
| + | H      | 1.  | 2.2 | 1.2 | 1.2 | .1  | .15 | .15 |
|   | CBEAM  | 4   | 4   | 4   | 5   | 0.  | 1.  | 0.  |
|   | PBEAML | 5   | 1   |     | I   |     |     |     |
| + | I      |     |     |     |     |     |     |     |
| + | I      | 2.2 | 1.2 | 1.2 | .1  | .15 | .15 | YES |
| + | J      |     |     |     |     |     |     |     |
| + | J      | 1.  | 2.  | 1.  | 1.  | .1  | .15 | .15 |
|   | CBEAM  | 5   | 5   | 5   | 6   | 0.  | 1.  | 0.  |

\$ Referenced Material Records

\$ Material Record : aluminum

\$ Description of Material : Date: 13-Feb-98

Time: 15:51:15

MAT1 1 1.+7 .3

\$ Nodes of the Entire Model

GRID 1 0. 0. 0.

GRID 2 10. 0. 0.

GRID 3 20. 0. 0.

GRID 4 30. 0. 0.

GRID 5 39.9999 0. 0.

GRID 6 50. 0. 0.

\$ Loads for Load Case : Default

SPCADD 2 1

LOAD 2 1. 1. 1

\$ Displacement Constraints of Load Set : cantilever

SPC1 1 123456 1

\$ Nodal Forces of Load Set : load

FORCE 1 6 0 1000. 0. -1. 0.

\$ Referenced Coordinate Frames

ENDDATA d87ffb7a

## SUBMITTING THE INPUT FILE FOR MSC/NASTRAN and MSC/PATRAN USERS:

11. Submit the input file to MSC/NASTRAN for analysis by finding an available UNIX shell window. At the command prompt enter **nastran pbeam.bdf scr=yes**. Monitor the run using the UNIX **ps** command.
  
12. Proceed with the Reverse Translation process, that is, importing the **pbeam.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

### ◆ Analysis

Action:

Read Output 2

Object:

Result Entities

Method:

Translate

Select Results File...

Selected Results File

pbeam.op2

OK

Apply

Before postprocessing the results, clear the LBC markers from the screen by selecting the following main menu icon:



**Reset Graphics**

When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

13. When the translation is complete and the Heartbeat turns green, bring up the **Results** form.

### ◆ Results

Action:

Create

Object:

Quick Plot

---

*Select Result Cases:*

**Default, Static Subcase**

*Select Fringe Result:*

**Displacements, Translational**

*Quantity:*

**Magnitude**

*Select Deformation Result:*

**Displacements, Translational**

**Apply**

**Figure: Fringe and Deformation plots of displacement.**

**Quit** MSC/PATRAN when you have completed this exercise.