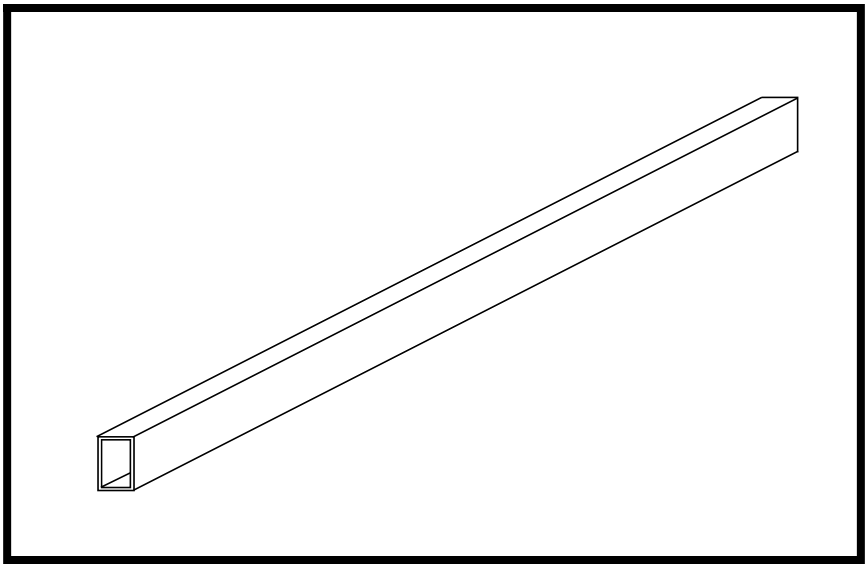
APPENDIX B

Linear Static Analysis of a Cantilever Beam Using Beam Library (SI Units)

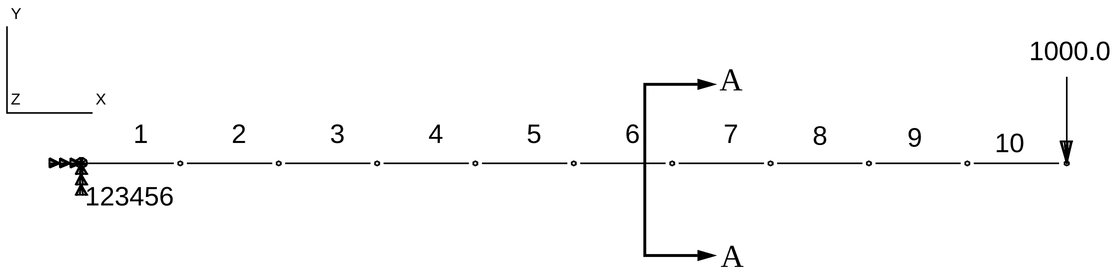


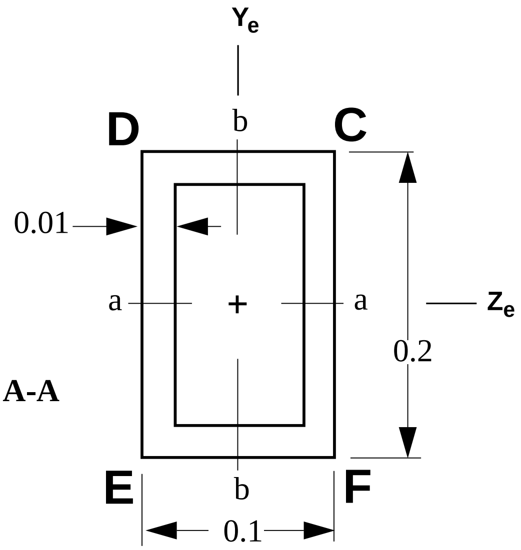
Objectives:

- Create a geometrical representation of a cantilever beam.
- Use the geometry model to define an MSC/NASTRAN analysis model comprised of CBAR elements.
- Prepare an MSC/NASTRAN input file for a linear static analysis.
- Visualize analysis results.

Model Description:

Below is a finite element representation of the beam structure shown on page 7-1. 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 condition 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.





View A-A

Elastic Modulus: **7.1e10 N/m^2**
Poisson Ratio: **0.3**
Density: **2.704e3 kg/m^3**
Area: **5.600e-3 m^2**
I_{aa}(I_{1-1}): **2.780e-5 m^4**
I_{bb}(I_{2-2}): **8.990e-6 m^4**
J: **2.090e-5 m^4**

Suggested Exercise Steps:

- Open a new database.
- Create a curve and mesh it with bar elements (CBAR). Use the meshing feature so that elements and nodes (GRID) will be generated automatically by MSC/PATRAN.
- Define material (MAT1) and element (PBAR) properties.
- Verify XY-orientation vectors for bar elements.
- Apply a fixed boundary constraint (SPC1) at one end of the beam and a transverse force to the free end of the beam (FORCE).
- Use the load and boundary condition sets to define a loadcase (SUBCASE).
- Prepare the model for a Linear Static analysis (SOL 101 and PARAMs).
- Generate and submit input file to the MSC/NASTRAN solver.
- Post-process results.
- Quit MSC/PATRAN.

Exercise Procedure:

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

File/New...

New Database Name:	**cantilever_beam**
OK	

In the New Model Preference form set the following:

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

2. Create a curve to define a geometrical representation of the beam.

◆**Geometry**

Action:	**Create**
Object:	**Curve**
Method:	**XYZ**

❑ **Auto Execute**

Vector Coordinates List:	**< 5, 0, 0 >**
Origin Coordinates List:	**[0, 0, 0]**

Apply

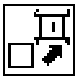
3. Discretize the geometry model with BAR2 elements. The element length is determined by the **Global Edge Length** parameter.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**0.5**
Element Topology:	**Bar2**
Curve List:	**Curve 1**

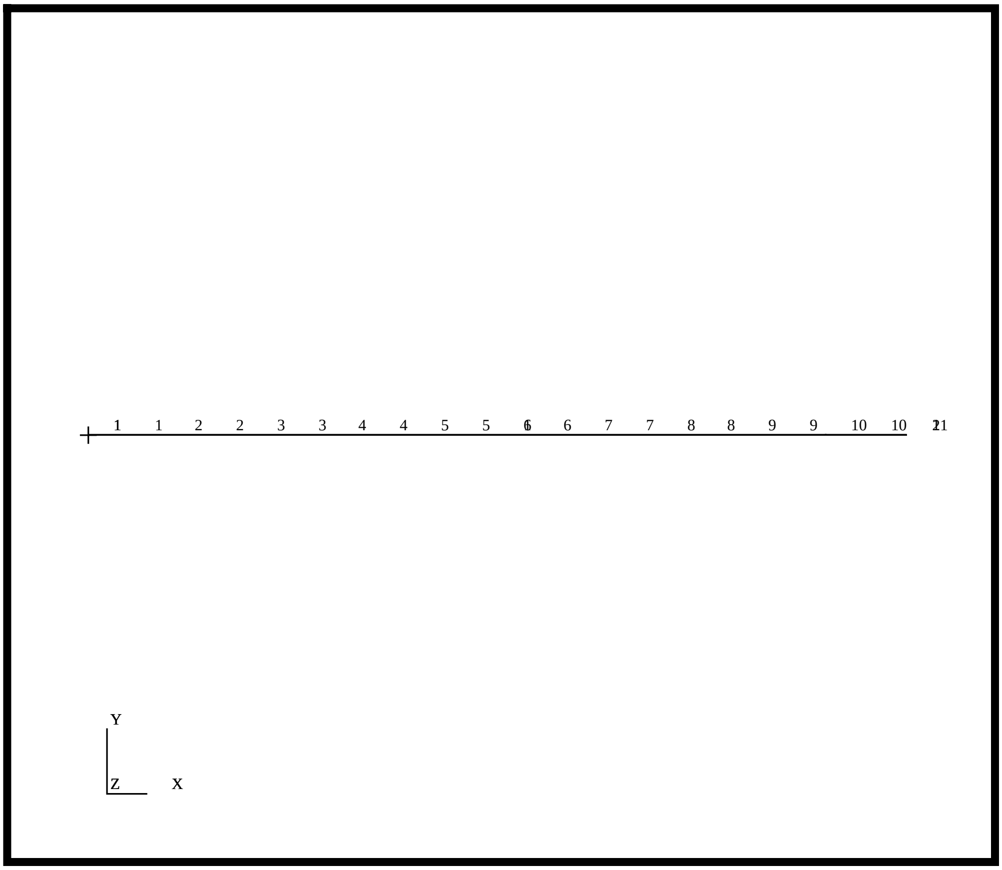
Apply

Show all entity labels by selecting the **Show Labels** icon on the **Top Menu Bar**



Show Labels

The completed model should appear as follows:



4. Define a material using the specified modulus of elasticity, Poisson ratio and density.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat_1**

Input Properties...

Constitutive Model:	**Linear Elastic**
Elastic Modulus =	**7.10e10**
Poisson Ratio =	**0.3**
Density =	**2.65e4**

Apply

The *Current Constitutive Models* form should appear as below:

Linear Elastic - [,,,,] - [Active]

Cancel

5. Define the properties of your beam model using the specified section properties data. Here is where the material defined in the previous operation is referenced. Be sure to specify the XY-orientation vector correctly. Also, remember to specify the stress recovery coefficients correctly. Otherwise, it will be impossible to recover bending stresses.

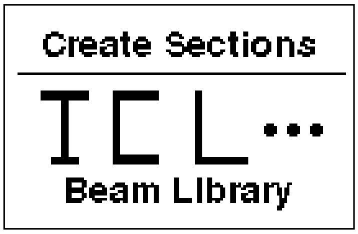
◆ **Properties**

Action:	**Create**
Object:	**1D**
Method:	**Beam**
Property Set Name:	**bar**

Input Properties...

Material Name:	**m:mat_1**
Bar Orientation:	**< 0., 1., 0. >**

Now, activate the Beam Library by clicking the icon shown below



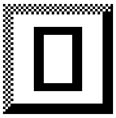
Beam Library (Create Section)

Create

Standard Shapes

New Section Name:	**sect_1**

Click on the icon shown below to access the **Box Section** menu.



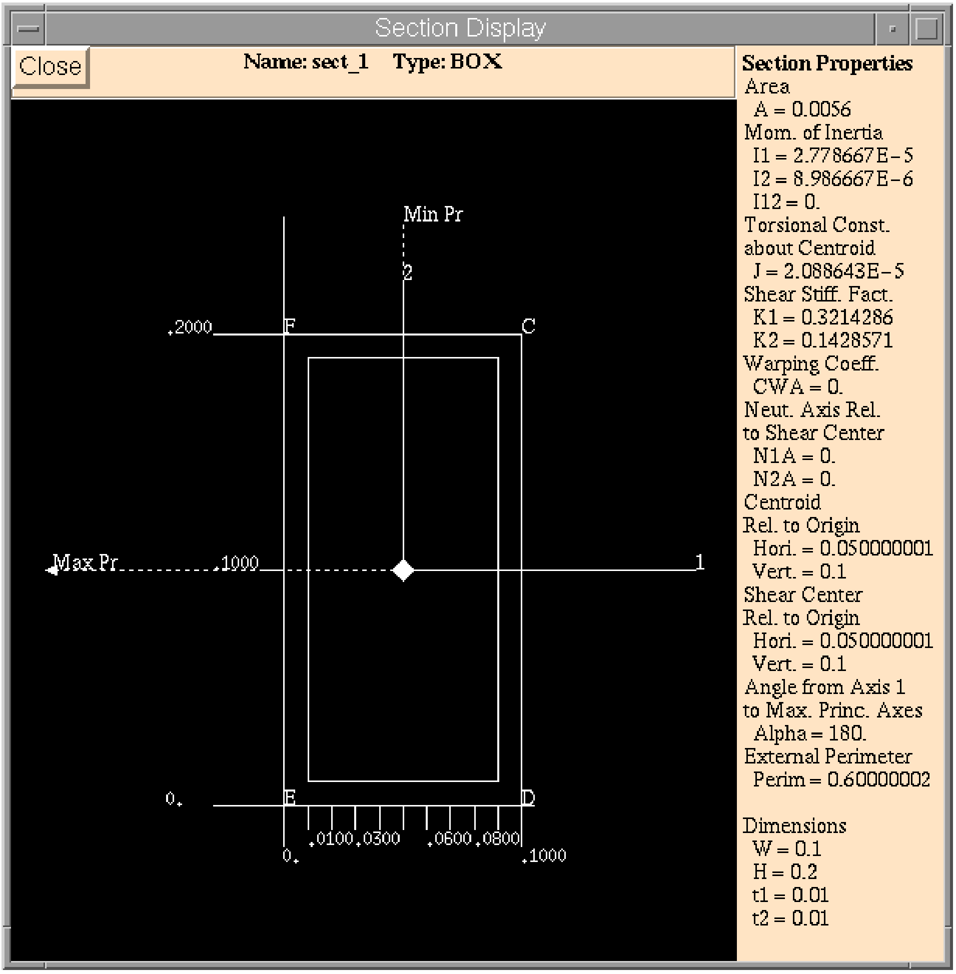
Box Section

The menu should appear in the right half of the window. Enter the following dimensions into it.

W =	**0.1**
H =	**0.2**
t1 =	**0.01**
t2 =	**0.01**

Calculate/Display

The window shown below should appear.



OK

OK

Select Members: **Curve 1**

Add

Apply

6.

7. Graphically assess the orientation vectors that are required on the CBAR entries in the MSC/NASTRAN input file. These vectors define the local XY-plane for each bar element. Since the element property created was applied to the geometry model instead of the analysis model, graphical display of respective attributes will appear on the geometry model by default. To display attributes such as the orientation vectors on our analysis model, we must change an option in *Display/Load/BC/Elem. Props...* The node labels may be deactivated for clarity.

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

■ **Show on FEM Only**

Beam Display: **3D: Full-Span+Offsets**

Apply

Cancel

Change the action in the **Element Properties** form to **Show**.

Action: **Show**

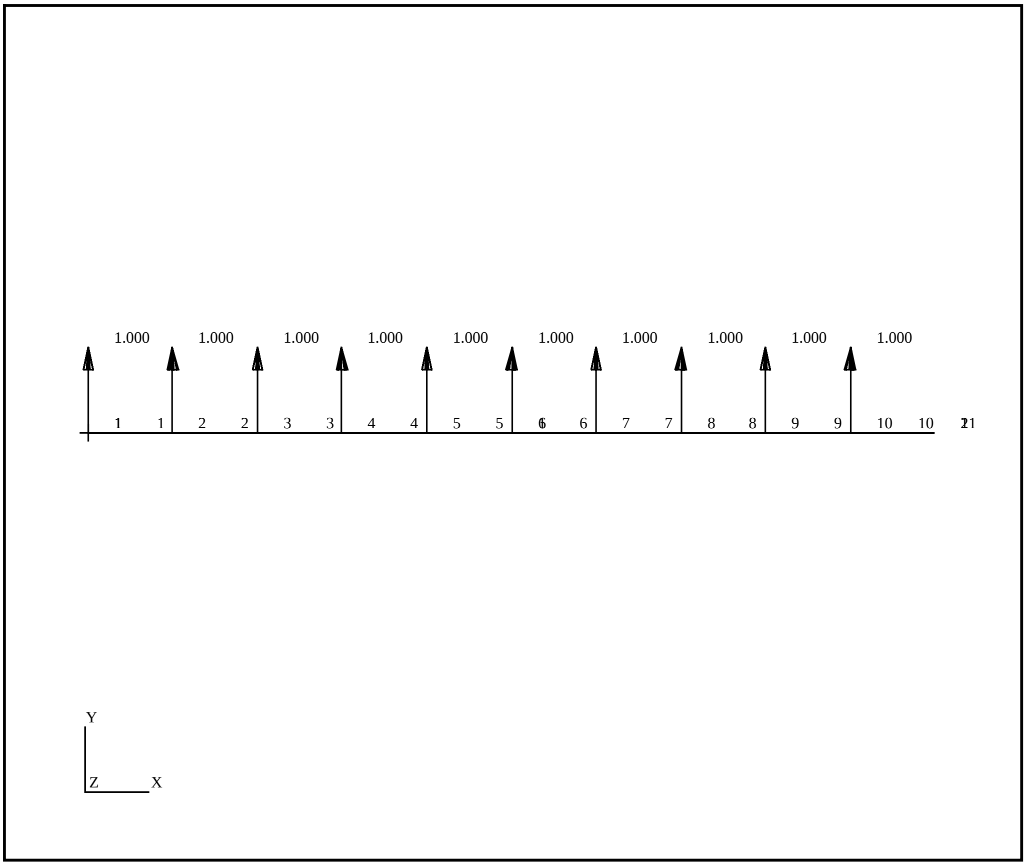
Existing Properties: **Definition of XY Plane**

Display Method: **Vector Plot**

Select Group: **default_group**

Apply

The display should appear as follows:



8. Reset the Functional Assignment Display back to geometry.

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

❑ **Show on FEM Only**

Apply

Cancel

8a. Define the cantilever boundary condition by creating displacement constraints and applying them to the geometry model.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**

New Set Name: **fixed**

Input Data...

Translation < T1 T2 T3 > **< 0, 0, 0 >**

Rotation <R1 R2 R3> **< 0, 0, 0 >**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

Select Geometric Entities: **Point 1**

Add

OK

Apply

9. The tip force which causes the beam to bend is defined as follows:

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **y_load**

Input Data...

Force <F1 F2 F3> **< , -1000, >**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

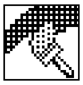
Select Geometric Entities: **Point 2**

Add

OK

Apply

Refresh the display by selecting the brush icon on the **Top Menu Bar**.

 **Refresh Graphics**

Create a load case which references the forces and boundary conditions that have been defined.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**sub_1**
Load Case Type:	**Static**
Assign/Prioritize Loads/BCs	
*(Click each selection until all Loads/BCs have one entry in the spreadsheet)**	**Displ_fixed** **Force_y_load**

*** REMINDER:** Make sure that the LBC Scale Factor column shows the proper value for each entry.

OK

Apply

10. For clarity, create a new group called **fem_only.** This group will contain only analysis model entities.

Group/Create...

New Group Name:	**fem_only**
■ **Make Current**	
■ **Unpost All Other Groups**	
Group Contents:	**Add All FEM**

Apply

Cancel

11. Again, for clarity, shrink the elements by 10%; this allows us to easily assess the element connectivities. Use the **Display/Finite Elements...** option.

Display/Finite Elements...

FEM Shrink:	**0.10**
Apply	
Cancel	

12. To display the load and boundary conditions on the analysis model, change the action in the **Loads/BCs** form to **Plot Markers**.

    12a. Recall that because the loads and boundary conditions you defined were applied to the geometry model, the Functional Assignment Display must be set to FEM.

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

■ **Show on FEM Only**

Apply

Cancel

    12b. Plot the load and boundary condition markers.

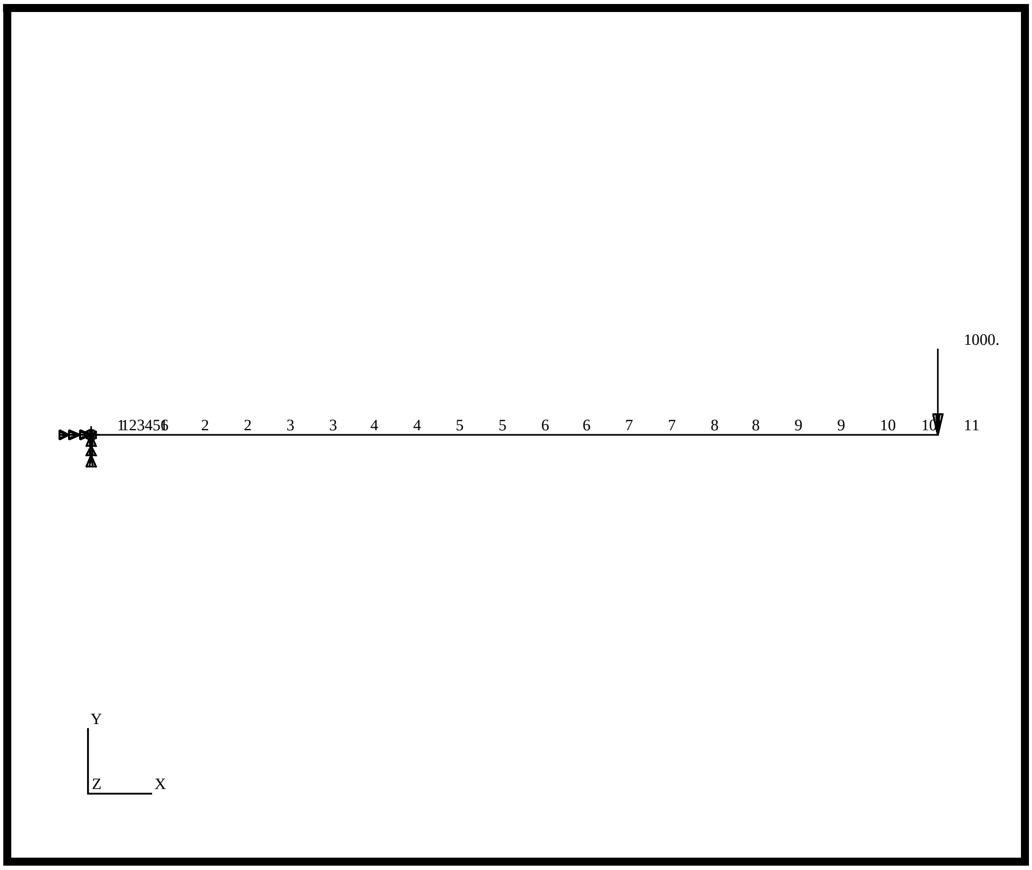
◆ **Loads/BCs**

Action:	**Plot Markers**

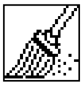
Select all sets in the *Assigned Load/BC Sets* box by highlighting them. Apply to the current group **fem_only**.

Assigned Load/BCs Sets:	**Displ_fixed** **Force_y_load**
Select Groups:	**fem_only**
Apply	

When you are done, you will see the load and boundary conditions displayed as follows::



Reset the display by selecting the broom icon on the **Top Menu Bar**.

 **Reset Graphics**

13. You are now ready to generate an input file for analysis.

Click on the **Analysis** radio button on the **Top Menu Bar** and complete the entries as shown below.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Analysis Deck**
Job Name:	**cantilever_beam**

Translation Parameters...

OUTPUT2 Format:	**Binary**
MSC/NASTRAN Version:	? ? ? set to current version **70**

OK

Solution Type...

Solution Type:	◆ **Linear Static**

Solution Parameters...

■ **Database Run**

■ **Automatic Constraints**

Data Deck Echo:	**Sorted**
Wt.- Mass Conversion =	**1.000** *(for SI units)*

OK

OK

Subcase Select...

Subcases For Solution Sequence:	**sub_1**
Subcases Selected:	**Default** *(click to deselect)*

OK

Apply

An input file named **cantilever_beam.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.

14. If all is well, you can submit the input file to MSC/NASTRAN for analysis. To do this, find an available xterm window and at the prompt enter:

    nastran cantilever_beam.bdf scr=yes

    Monitor the run using the UNIX **ps** command.

14a. When the run is completed, edit the **cantilever_beam.f06** file and search for the word **FATAL**. If none exists, search for the word **WARNING**. Determine whether or not existing WARNING messages indicate modeling errors.

14b. While still editing **cantilever_beam.f06**, search for the word

D I S P L A C E (spaces are necessary)

What is the y-component of the tip deflection vector?

disp Y = ____________

What is the %error of this deflection versus the theoretical tip deflection?

%error = ____________

Search for the word:

S T R E S S (spaces are necessary)

What is the maximum positive stress due to bending?

max. stress = ____________

What is the %error of this stress versus the theoretical maximum positive bending stress?

%error = ____________

15. Proceed with the Reverse Translation process, that is, importing the **cantilever_beam.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...

Filter

Selected Results File: *select the desired **.op2** file*

OK

Apply

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

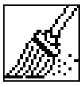
◆ **Results**

Action: **Create**

Object: **Quick Plot**

Choose the desired result case in the **Select Result Cases** list and select the result(s) in the **Select Fringe Result** list and/or in the **Select Deformation Result** list. And hit **Apply** to view the result(s) in the viewport.

If you wish to reset your display graphics to the state it was in before you began post-processing your model, remember to select the broom icon.



Reset Graphics

Quit MSC/PATRAN when you have completed this exercise.