LESSON 9

Linear Static Analysis of a Cantilever Beam (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.

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)

9-2

Model Description:

Below is a finite element representation of the beam structure shown on page 9-1. 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.



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:

Analysis Code:

▼ Dela

Analysis Type:

OK

♦ Default

MSC/NASTRAN

Structural

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

♦ Geometry	
Action:	

Object:	Curve
Method:	XYZ

Auto Execute

Vector Coordinates List:

Origin Coordinates List:

Apply

<	5,	0,	0	>	

0]

[0, 0,

Create

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:

Apply

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

Curve 1





The completed model should appear as follows:

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



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. >
(scroll down using scroll bar)	
Area:	0.0056
[Inertia 1,1]:	(Enter inertia about 1-1)
[Inertia 2,2]:	(Enter inertia about 2-2)
(scroll down using scroll bar)	
[Torsional Constant]:	2.090e-5
(scroll down using scroll bar)	

[Y of Point C]:	0.10
[Z of Point C]:	0.05
[Y of Point D]:	0.10
[Z of Point D]:	-0.05
[Y of Point E]:	-0.1
[Z of Point E]:	-0.05
[Y of Point F]:	(Enter Y-coord. of point F)
[Z of Point F]:	(Enter Z-coord. of point F)
ОК	
Select Members:	Curve 1
Add	

6. 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

Apply	
Cancel	

9-8

Apply

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:



7. Reset the Functional Assignment Display back to geometry.

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

Show on FEM Only

Apply Cancel

7a. Define the cantilever boundary condition by creating displacement constraints and applying them to the geometry model.

♦ Loads/BCs

Action:

Create

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5) 9-9



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



9-10

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)

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:

Load Case Name:

Load Case Type:

Assign/Prioritize Loads/BCs

(Click each selection until all Loads/BCs have one entry in the spreadsheet)*

Displ fixed
I –
Force_y_load

Create

sub_1

Static

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

OK	
Apply	

9. 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

MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5) 9-11

Apply	
Cancel	

10. 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...

Apply	
Cancel	

FEM Shrink:

0.10

- 11. To display the load and boundary conditions on the analysis model, change the action in the **Loads/BCs** form to **Plot Markers**.
 - 11a. 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

11b. 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	
fem_only	

Select Groups:

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**.



12. 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.





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.

13. 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.

9-14 MSC/NASTRAN 120 Exercise Workbook - Version 70 (MSC/PATRAN 7.5)

- 13a. 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.
- 13b. While still editing **cantilever_beam.f06**, search for the word

DISPLACE (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 =

14. 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:



Action:

Object:

Method:

Read Output2
Result Entities
Translate

Select Res	sults File
Filter	
Selected F	Results File:
OK	
Apply	

select the desired .op2 file

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.



Quit MSC/PATRAN when you have completed this exercise.