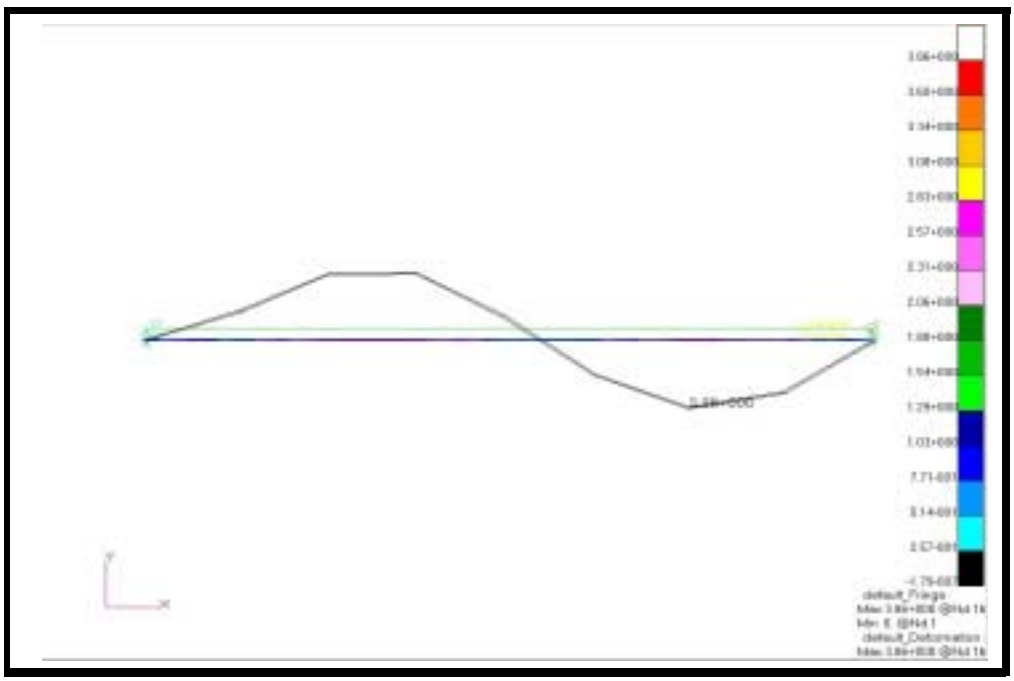
LESSON 3

Modal Analysis of a Cantilever Beam



Objectives:

- Create the cantilever beam model as in Lesson 01.
- Analyze model using MSC.Marc.
- Compare results to hand solution.

Exercise Description:

In this Analysis you will extract the first three modes of a simply supported - fixed beam. You will compare these results to theoretical conditions. Next, you will apply an axial load to the end of the beam. This is analogous to stretching a guitar string. The result is the amount the frequency should increase. You will then compare the results of the preloaded and unloaded structure. This entire procedure will be done in a single analysis using three analysis Load Steps.

The geometric and material properties for the cantilever beam. are:

$E = 30E6\ N/m^2$

$\nu = 0.30$

$\rho = 0.00074$

Exercise Procedure:

1. Create a new database named **natural_frequency.db.**

File/New ...

New Database Name: **natural_frequency.db**

OK

In the New Model Preference form set the *Analysis Code* to **MSC.Marc.**

Tolerance: ● **Based on Model**

Analysis Code: **MSC.Marc**

Analysis Type: **Structural**

OK

2. Import the old database. Use the cantilever beam model from the first part of this exercise.

File/Import ...

Object: **Model**

Source: **MSC.Patran DB**

Import File: **cantilever_beam**

This will be the old database just created.

Apply

Close the summary form by selecting “OK.”

OK

3. Now graphically display only the cantilever beam.

Group/Post...

Selected Groups to Post: **cantilever_beam**

Apply

Cancel

4. Apply a simple support at the right end.

The left end of the beam is fixed in all active degrees of freedom and was already defined back in *Exercise 1 - Build a Cantilever Beam*. Now add a simple support to the right side of the beam so that it cannot deflect in the vertical direction as follows:

■ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Method:	**Nodal**
New Set Name:	**simply_supported**

Input Data...

Translations <T1,T2,T3>:	**< , 0 >**

OK

Select Application Region...

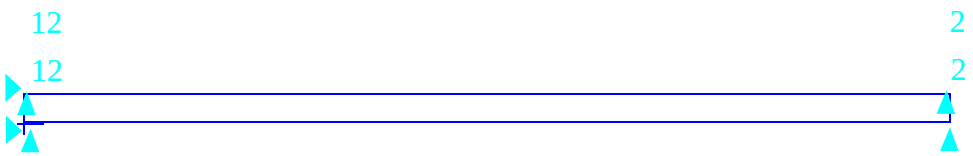
Geometry Filter:	● **Geometry**
Select Geometric Entities:	**Point 3 4**

Add

OK

Apply

Figure 3.1: The beam with fixed end and Boundary Condition.



5. Create two new Load Cases

Create a new load case for the beam constraints only and another with the constraints and the new tension load that will be created next.

■ **Load Cases**

Action:	**Create**

Load Case Name: **Modal_Constraints**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet: **Displ_fixed** **Displ_simply_supported**

OK

Apply

Load Case Name: **Tension_Load**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet: **Displ_fixed** **Displ_simply_supported**

OK

Apply

6. Adding a tip tension Load

Add a 20,000 lb tip load to the model to impose tension in the beam. The *Tension_Load* load case is the current load case since it was the last created. This load will be placed in this load case.

■ **Loads/BCs**

Action: **Plot Markers**

Assigned Load/BC Sets: Select all the LBC's in this listbox

Select Groups: **cantilever_beam**

Apply

Action: **Create**

Object: **Force**

Method: **Nodal**

New Set Name: **tension_load**

Input Data...

Force <F1,F2,F3>: **< 10000, , >**

OK

Select Application Region...

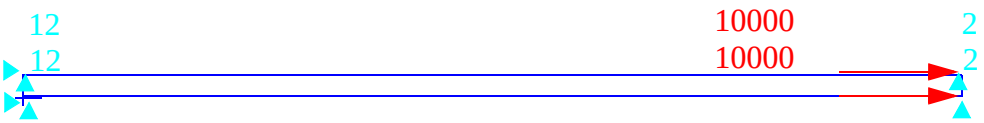
Select Geometric Entities: **Point 3 4**

Add

OK

Apply

Figure 3.2: The current model, including tension load.



7. Create the Analysis load steps

We need to create three **Load Steps**. Two for modal extraction before and after the preload is applied and another to apply the preload.

■ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Job Name: **normal_modes**

Load Step Creation...

Job Step Name: **Modal Step1**

Solution Type: **Normal Modes**

Solution Parameters...

Number of Modes: **3**

Highest Frequency: **300**

OK

Select Load Case...

Available Load Cases: **Modal_Constraints**

OK

Apply

Available Job Steps: **Modal Step1**

Job Step Name: **Modal Step2**

Apply

Job Step Name: **Tension Preload Step**

Solution Type: **Static**

Solution Parameters...

Load Increment Parameters...

Increment Type: **Fixed**

Number of Increments: **5**

OK

OK

Select Load Case...

Available Load Cases: **Tension_Load**

OK

Apply

Cancel

8. Submit the Analysis

Now select the steps in the proper order for the Analysis.

Load Step Selection...

Existing Job Steps: **Modal Step1**
Tension Preload Step
Modal Step2

Selected Job Steps: **Default Static Step**

OK

Apply

In the listed order, the three selected Job Steps should now be in place of where the **Default Static Step** was.

9. Monitor the Analysis

MSC.Marc creates several files that you can use to monitor your job and verify that the analysis has run correctly. The `.log` file is an ASCII file that contains a summary of all load increments. Upon completion of the analysis, the `.out` file provides you with a summary of any job errors. Another useful ASCII file is the `.sts` file. This file contains a summary of job information including step number, number of increments, number of iterations, total time of step, and time of a given increment. You can view these files during or after a job has completed using standard UNIX or Windows editors or use the MSC.Analysis Manager or the **Monitor** action in the **Analysis** application (this is the most convenient method).

Action:	**Monitor**
Object:	**Job**
View Status File...	

When the end of this file has the following line, you know that the job has completed successfully: Job ends with exit number : 3004

10. Read the Results

When the analysis job is finished, attach the results file to be able to postprocess as before.

■ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**
Method:	**Attach**
Select Results File...	**normal_modes.t16**
OK	
Apply	

11. Plot the mode shapes

Use MSC.Patran to postprocess the results of the modal analysis. There will be a number of increments from the analysis for the first mode shapes and the titles should have the frequencies listed. The preload analysis itself will contain five more increments (or result

cases) each of which build up the tension load until the maximum load is reached. Then there are three more increments corresponding to the modes after the tension load is applied. Plot the mode shapes.

■ **Results**

Action:	**Create**
Object:	**Quick Plot**
Selected Results Case:	**...A1..., Mode=2, ...**
Selected Fringe Results:	**Displacement, Translation**
Selected Deformation Results:	**Displacement, Translation**

Apply

Change the Display Properties for results. Click on the Deform Attributes icon.

Scale Interpretation:	● **Model Scale**
Scale Factor:	**0.1**

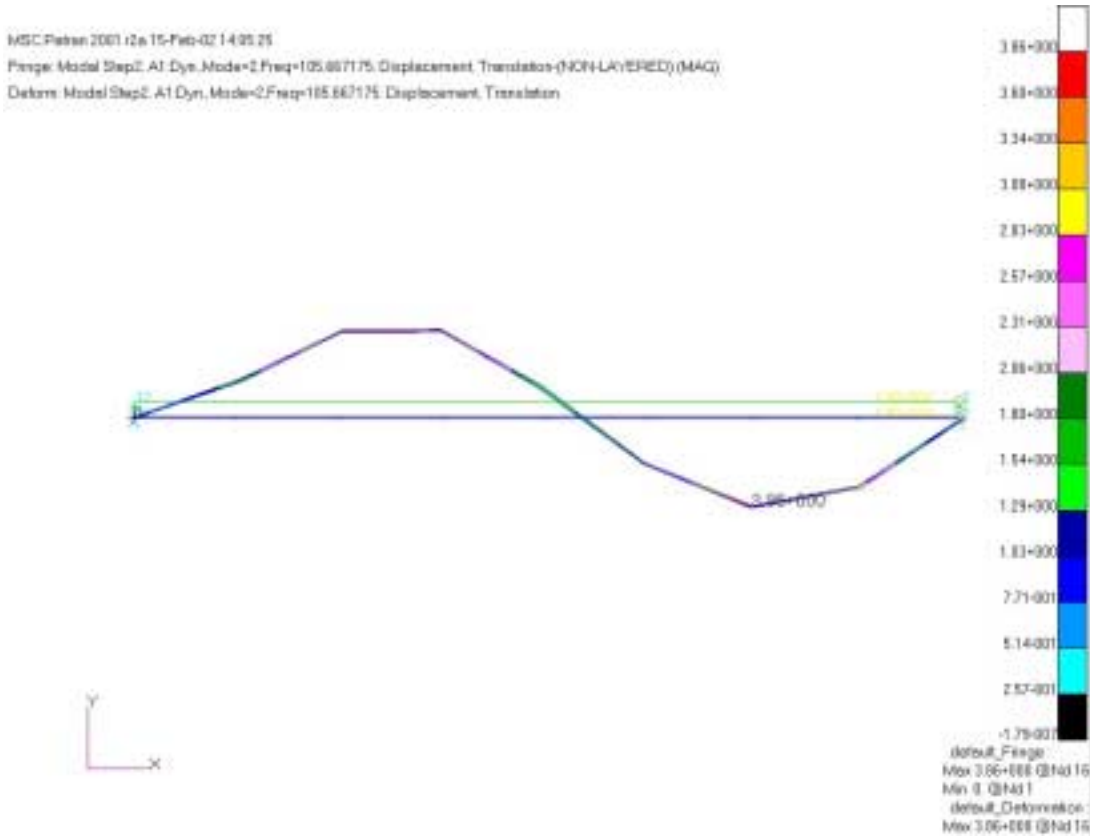
Apply

Choose the Fit View icon, or use the toolbar menu option under **Viewing/Fit View**.

Repeat this step for as many modes as you wish to view.

The following plot should appear, in **Figure 3.3**.

Figure 3.3: Mode shapes plotted from the Analysis.



Close the database and quit PATRAN.

This concludes this exercise.