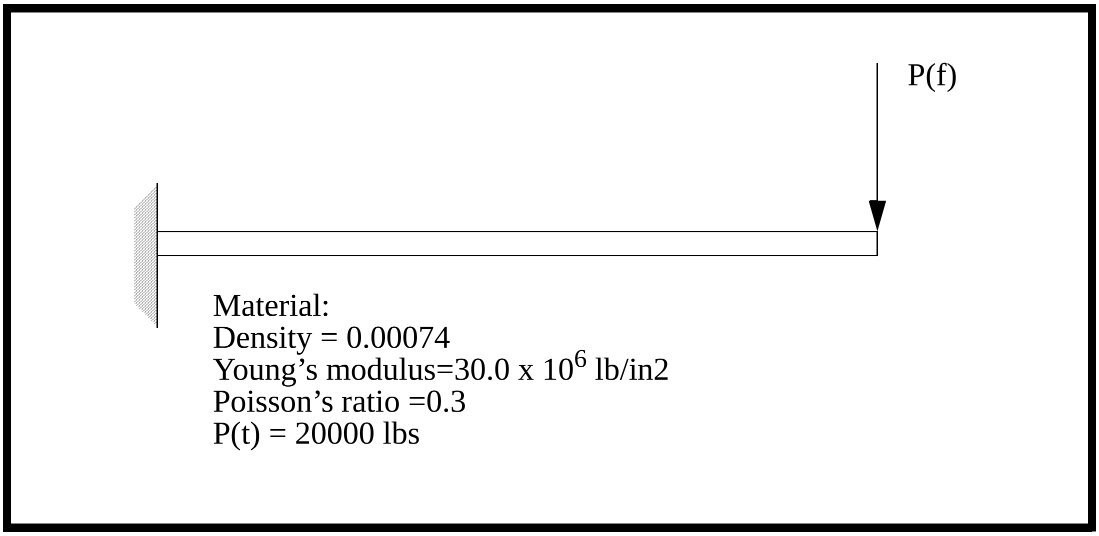
LESSON 21

Frequency Response Analysis of a Cantilever Beam

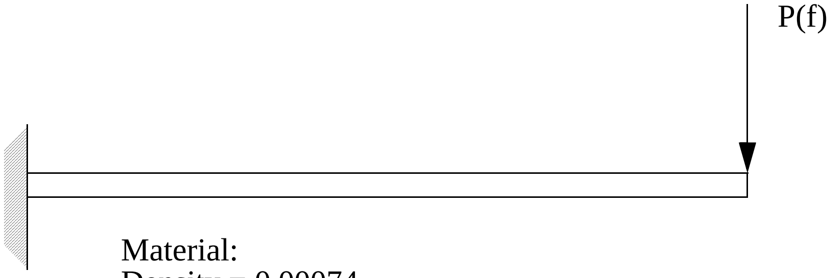


Objectives:

- Excite a cantilever beam with a beam-end load at various frequencies, set up inMSC.Marc.
- First perform a modal analysis, and compare the results with the static solution.

Model Description:

In this exercise, you will excite the cantilever beam with a load at the end of the beam at various frequencies (known also as a harmonic analysis). This characterizes its vibrational qualities. First a modal analysis will be done to determine the natural frequencies of the cantilever beam which will tell us approximately where we can expect to see large responses due to excitation at resonant frequencies. The results are also compared to the static solution at zero hertz.



Material:
Density = 0.00074
Young's modulus=30.0 x 10^6 lb/in2
Poisson's ratio =0.3
P(t) = 20000 lbs

Exercise Procedure:

1. Open a new database called **transient_dynamic_beam.**

File/New ...

New Database Name: **beam_freq_resp**

OK

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC/PATRAN.

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. Create a Load Case in which we will place all the Loads and Boundary Conditions of this Analysis.

■ **Load Cases**

Action: **Create**

Load Case Name: **Harmonic_Loads**

Assign/Prioritize Loads/BCs

Select individual Loads/BCs: **Displ_fixed**

OK

Apply

5. Create the harmonic Load

A 200 lb load is to be placed at the end of the beam.

■ **Loads/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **harmonic_force**

Input Data...

Force <F1, F2, F3>: **<, -100 >**

OK

Select Application Region...

Geometry Filter: ● **Geometry**

Select Geometric Entities: **point 3 4**

Add

OK

Apply

6. Set up the Load Steps for Analysis.

Two analyses will be done. The first is a modal analysis to determine the natural frequencies. The second is the frequency response analysis. The modal analysis needs a normal modes **Load Step**. The frequency response analysis will be done in two **Load Steps** (which is not really necessary but done to show the static response). The first step is the static solution with no harmonic excitation and the second step is the harmonic excitation with frequencies from zero to 100 Hz. Default values will be used for solution parameters for all the **Load Steps**.

■ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**

Load Step Creation...

Job Step Name:	**Modal Step**
Solution Type:	**Normal Modes**

Select Load Case...

Available Load Case:	**Default**

OK

Apply

Job Step Name:	**Static Step**
Solution Type:	**Static**

Select Load Case...

Available Load Case:	**Harmonic_Loads**

OK

Apply

Job Step Name:	**Harmonic Step**
Solution Type:	**Frequency Response**

Select Load Case...

Available Load Case: **Harmonic_Loads**

OK

Apply

Cancel

7. Submit the modal Analysis. Give the Analysis a jobname and select the proper Load Step.

Job Name: **modal**

Load Step Selection...

First select *Impulse Step*. Then deselect *Default Static Step* from the **Selected Job Steps** form.

Existing Job Steps: **Modal Step**

Selected Job Steps: **Default Static Step**

OK

Apply

Again, you will need to monitor the analysis for job completion. After the job starts to run, MSC.Marc creates several files that can be used to monitor the job and verify that the analysis has run correctly. The ***modal.log*** is an ASCII file which contains Element, Loads & Boundary Conditions, Material Translation, Step Control parameters, Equilibrium and Error information. When the job completes, this file contains an *Analysis Summary* which summarizes the error and iteration information. Another useful ASCII file is the ***modal.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. The ***modal.out*** file contains a summary of any job errors. These files can be viewed during or after a job has completed. A more convenient method might be to use the **Analysis** application, **Monitor**.

Action: **Monitor**

Object: **Job**

View Status File...

After the job has finished, a successful completion will end with the line: Job ends with exit number: 3004

8. Submit the frequency response Analysis.

When the above job is done, submit the frequency response Analysis.

Job Name:	**freq_resp**
Load Step Selection...	
Existing Job Steps:	**Static Step** **Harmonic Step**
Selected Job Steps:	**Modal Step**
OK	
Apply	

Again, Monitor the job if you wish.

9. Read in the results when the two analysis jobs are finished.

◼ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**
Method:	**Attach**
Available Jobs:	**modal**
Select Results File...	**modal.t16**
OK	
Apply	
Available Jobs:	**freq_resp**
Select Results File...	**freq_resp.t16**
OK	
Apply	

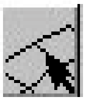
10. Plotting the tip deflection with frequency.
Plot the Y displacement with respect to frequency for the right top node at the free end. First note the result case names for the modal analysis. They should have the frequencies listed. The first frequency is at around 6.5 Hz and the second is around 40 Hz. We

should expect higher responses at these frequencies. This is dependent, of course, at which node point we are inspecting.

■ **Results**

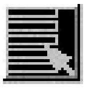
Action:	**Create**
Object:	**Graph**

Select the **Target Entity** icon

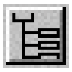


Target Entity:	**Nodes**
Nodes:	**Node 18**

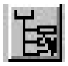
Go to the **Select Results** form



Select **View Subcases** icon, highlight the **Harmonic Step** Case, then choose the **Select Subcases** icon.



Select Result Cases:	**Harmonic Step, ...subcases**



Filter

Apply

Close

Y:	**Result**

Select Y Result: **Displacement, Translation**

Quantity: **Y Component**

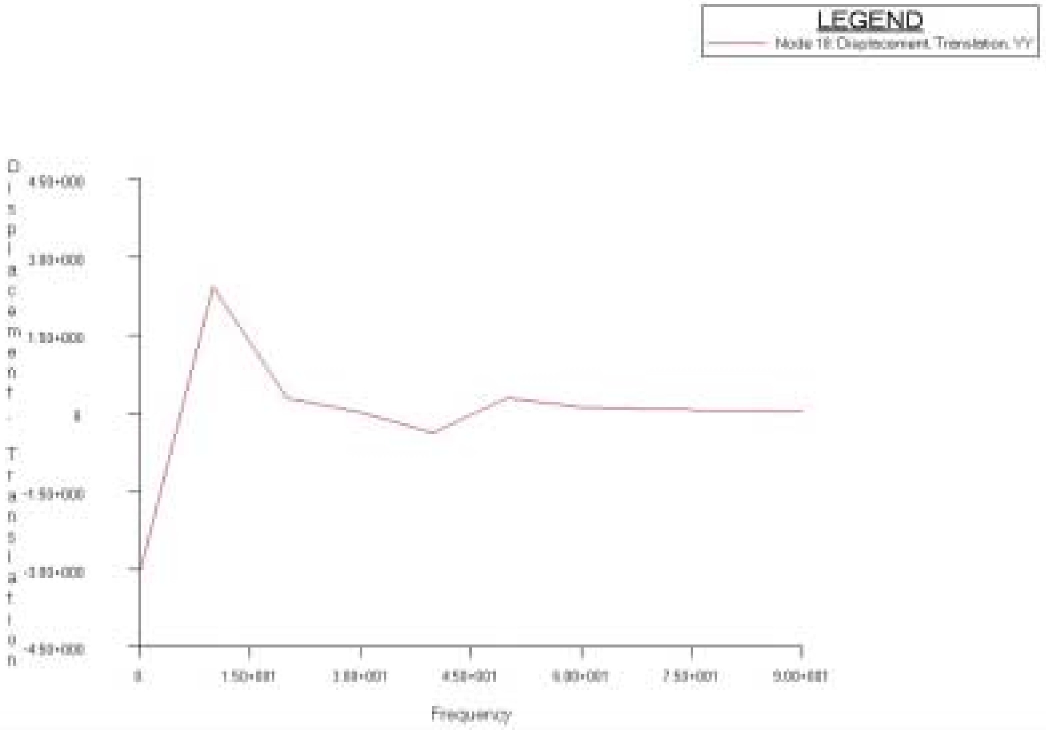
X: **Global Variable**

Global Variable: **Frequency**

Apply

The results are shown below, in Figure 21.2. As expected, there are peak responses at around 6.5 and 40 Hz.

Figure 21.2 - Result of the Analysis



The results are fairly coarse. A better frequency resolution would capture the response around the resonant frequencies much better. By default, only 10 frequencies were analyzed between 0 and 100 Hz. This can be changed in the **Solution Parameters** form. You can also set up multiple harmonic **Load Steps** if you need to zoom in around particular frequencies yet need more coarse resolution at other frequencies.

Close the database and quit PATRAN.

This concludes the exercise**.**