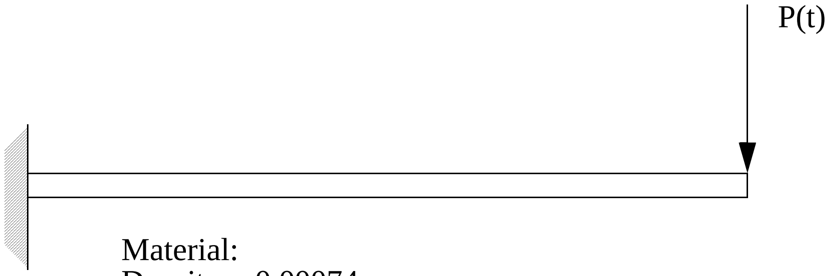
LESSON 20

Transient Dynamic Analysis of a Cantilever Beam



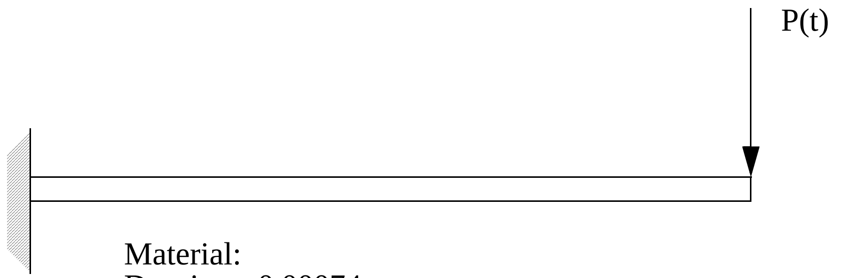
Material:
Density = 0.00074
Young's modulus=30.0 x 10^6 lb/in2
Poisson's ratio =0.3
P(t) = 6000 lbs

Objectives:

- Run a direct transient dynamic analysis set up in MSC.Marc.
- Analyze with and without damping analysis, and with a contact interference.

Model Description:

In this exercise, you will apply an impulse load on the free end of a cantilever beam. The loading will be a force defined as a function of time; therefore you will need to define a nonspatial field. You will run the analysis as a direct transient analysis, first without damping, then with damping, and finally, with a contact interference.



Material:
Density = 0.00074
Young’s modulus=30.0 x 10^6 lb/in2
Poisson’s ratio =0.3
P(t) = 6000 lbs

Exercise Procedure:

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

File/New ...

New Database Name:	**transient_dynamic_beam**

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 the time history

Create the time history for the impulse loading.

■ **Fields**

Action: **Create**

Object: **Non Spatial**

Method: **Tabular Input**

Field Name: **impulse**

Table Definition: **■ Time (t)**

Input Data...

Table 20.1: Fill in the table as shown below in the **Input Data** form. You must click in the cell before you can start entering data in the databox above the spreadsheet.

Time (t)	Value
0.0	0.0
0.015	1.0
0.03	0.0
1.0	0.0

OK

Apply

5. Create a time dependent Load Case

Up to this point we have dealt with only static or pseudo-static loading in general. This analysis however, requires a time dependent loading to be defined.

◼ **Load Cases**

Action: **Create**

Load Case Name: **Impulse_Load**

Load Case Type: **◆ Time Dependent**

Assign/Prioritize Loads/BCs

Select individual Loads/BCs: **Displ_fixed**

OK

Apply

6. Create the dynamic Load

A 20,000 lb load is to be placed at the end of the beam.

◼ **Loads/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **impulse_force**

Input Data...

Force <F1, F2, F3>: **<, -3000 >**

This associates the load with the time dependency. Click in this databox to activate it then select the field from the listbox containing **Time/Freq Dependent Fields**.

Time/Freq. Dependence: **f:impulse**

OK

Select Application Region...

Geometry Filter: **● Geometry**

Select Geometric Entities: **point 3 4**

Add

OK

Apply

The displayed value of the load will be zero. This is because the force value is multiplied by the first value in the field, which is zero.

7. Set up the model for analysis.

■ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Job Name:	**impulse**

Load Step Creation...

Job Step Name:	**Impulse Step**
Solution Type:	**Transient Dynamic**

Solution Parameters...

Linearity:	**Linear**

Load Increment Parameters...

Increment Type:	**Fixed**
Time Step Size:	**0.005**
Total Time:	**1.0**

OK

OK

Select Load Case...

Available Load Case:	**Impulse_Load**

OK

Apply

Cancel

Now select the steps in the **Analysis** form.

Load Step Selection...

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

Existing Job Steps:	**Impulse 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 ***impulse.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 ***impulse.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 ***impulse.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. Read in the results when analysis job is finished.

■ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**

Method: **Attach**

Available Jobs: **impulse**

Select Results File... **impulse.t16**

OK

Apply

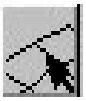
9. Plot the tip deflection with time. Plot the Y-displacement with respect to time for the right top node at the free end.

◆ **Results**

Action: **Create**

Object: **Graph**

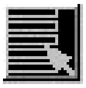
Select the **Target Entity** icon



Target Entity: **Nodes**

Nodes: **Node 18**

Go to the **Select Results** form



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

Filter

Apply

Close

Y: **Result**

Select Y Result: **Displacement, Translation**

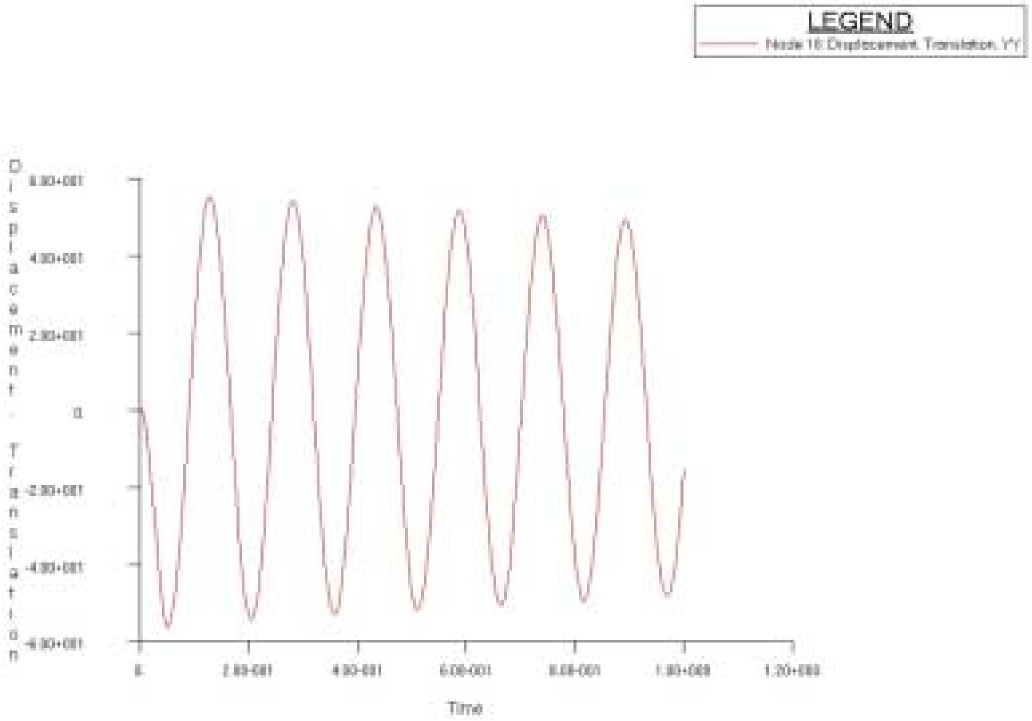
Quantity: **Y Component**

X:	**Global Variable**
Global Variable:	**Time**

Apply

The results are shown below, in Figure 20.2. The maximum displacement of a sudden load placed on the end of a beam in a dynamic solution (using small strain, small displacement theory), should be about twice that of the static solution. Since this is an impact load of very short duration, it does not even attain the deflection of the static solution

Figure 20.2 - Result of the Analysis



10. What about damping?

This model is undamped. The oscillation would be expected to continue for quite awhile which is obviously unrealistic. Add a material damping constitutive model to the material *steel*.

■ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**

Existing Materials:	**steel**
Input Properties...	
Constitutive Model:	**Damping**
Stiffness Matrix Multiplier:	**1e-3**
OK	
Apply	

11. Rerun Analysis with damping

■ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Available Jobs:	**impulse**
Job Name:	**impulse_damp**
Apply	

Again, Monitor the job if you wish.

12. Read and Plot Results with damping

Here, **Step 12** is repeated, which will read the results, this time selecting the new Job created, and then plot the displacement with respect to time for the right top node at the free end.

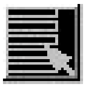
Action:	**Read Results**
Object:	**Result Entities**
Method:	**Attach**
Available Jobs:	**impulse_damp**
Select Results File...	**impulse_damp.t16**
OK	
Apply	

◆ **Results**

Action: **Create**

Object: **Graph**

Press the icon to select to select to the **Results Cases**. Node 18 should still be selected.



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

Clear

Filter Method: **String**

Filter String: ***A2***

Filter

Apply

Close

Y: **Result**

Select Y Result: **Displacement, Translation**

Quantity: **Y Component**

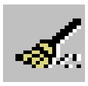
X: **Global Variable**

Global Variable: **Time**

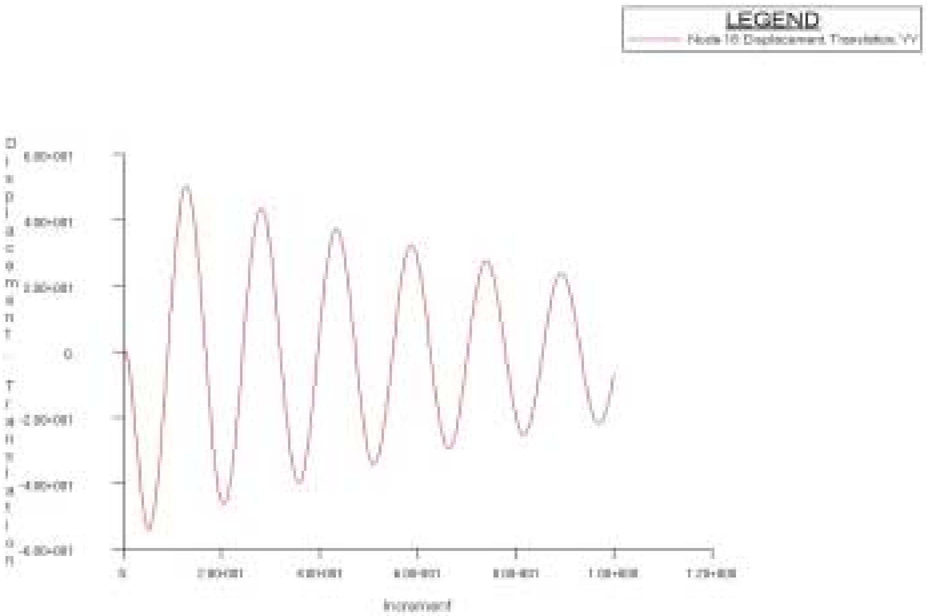
Apply

Figure 20.3: The deflection is not being attenuated over time in a more significant matter.

Note: To remove the graph, press the *Reset Graphics* icon.



13. Create a Contact interference



Now graphically display the cantilever beam and the interference geometry.

Group/Post...

Selected Group to Post: | cantilever_beam rigid_body1

Apply

Cancel

Define the deformable and rigid Contact bodies.

■ **Loads/BCs**

Action: | **Create**

Object: | **Contact**

Type: | **Element Uniform**

Option: | **Deformable Body**

New Set Name: | beam

Target Element Type: | **2D**

Select Application Region...

Select Surfaces: **Surface 1**

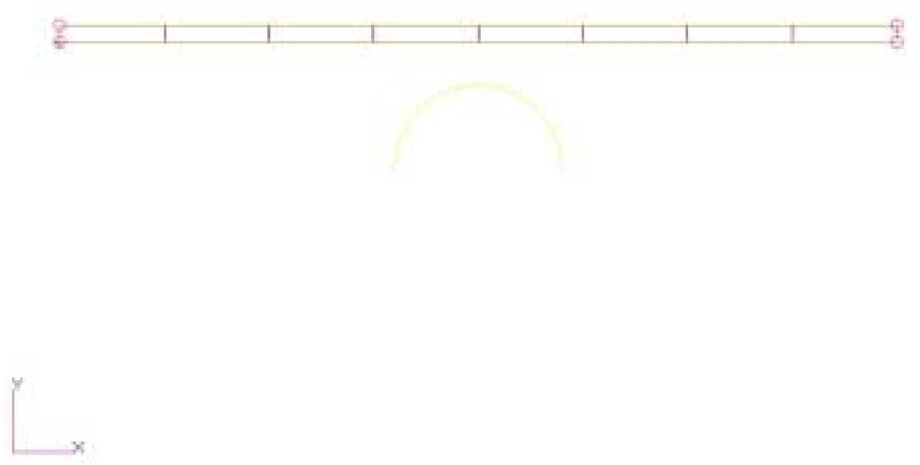
Add

OK

Apply

That does it for the deformable body. You should see some small round magenta circles in the corners of **Surface 1** that defines the deformable body, as in **Figure 20.4** below.

Figure 20.4: The deformable body contact definition.



Now, continue with the rigid body.

Option:	**Rigid Body**
New Set Name:	rigid_stop
Target Element Type:	**1D**
Select Application Region...	
Select Curves:	Curve 1
Add	
OK	
Apply	

Tic marks should appear along the arc pointing inward. If they are pointing the wrong way, just turn **ON** the **Flip contact Side** toggle on the **Input Data...** form and recreate the contact body. Now plot all the LBC markers in this load case.

■ **Loads/BCs**

Action: **Plot Markers**

Assigned Load/BC Sets: **Conta_beam**
Conta_rigid_stop
Displ_fixed
Force_impulse_force

Select Groups: **cantilever_beam**
rigid_body1

Apply

14. Deactivate the damping.

Analyze this model undamped. Turn the damping constitutive model off for the material *steel*.

■ **Materials**

Action: **Create**

Object: **Isotropic**

Method: **Manual Input**

Existing Materials: **steel**

Cancel

(Close the **Input Properties** form that automatically appears.)

Change Material Status...

Active Constitutive Models: Select to make inactive, **Damping**

Apply

Cancel

Apply

15. Set up the model and submit the Analysis.

Modify the Load Step for contact Analysis.

■ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method:	**Full Run**
Available Jobs:	**impulse**
Job Name:	**impulse_contact**
Load Step Creation...	
Available Job Steps:	**Impulse Step**
Solution Type:	**Transient Dynamic**
Solution Parameters...	
Linearity:	**NonLinear**
NonLinear Geometric Effects:	**Large Displ.(Updated Lagr.) /Small Strains**
Load Increment Parameters...	
Increment Type:	**Fixed**
Time Step Size:	**0.005**
Total Time:	**1.0**
OK	
OK	
Select Load Case...	
Available Load Case:	**Impulse_Load**
OK	
Apply	

This modifies the existing **Load Step**. Answer **YES** to overwrite

Cancel

Load Step Selection...

Make sure only **Impulse Step** is selected.

OK

Apply

16. Read and Plot Results with contact.

Again, **Step 12** is repeated, which will read the results. Select the new Job created, and plot the displacement with respect to time for the right top node at the free end.

Action: **Read Results**

Object: **Result Entities**

Method: **Attach**

Available Jobs: **impulse_contact**

Select Results File... **impulse_contact.t16**

OK

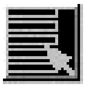
Apply

◆ **Results**

Action: **Create**

Object: **Graph**

Press the icon to select to select to the **Results Cases**. Node 18 should still be selected.



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

Clear

Filter Method: **String**

Filter String: ***A3***

Filter

Apply

Close

Y: **Result**

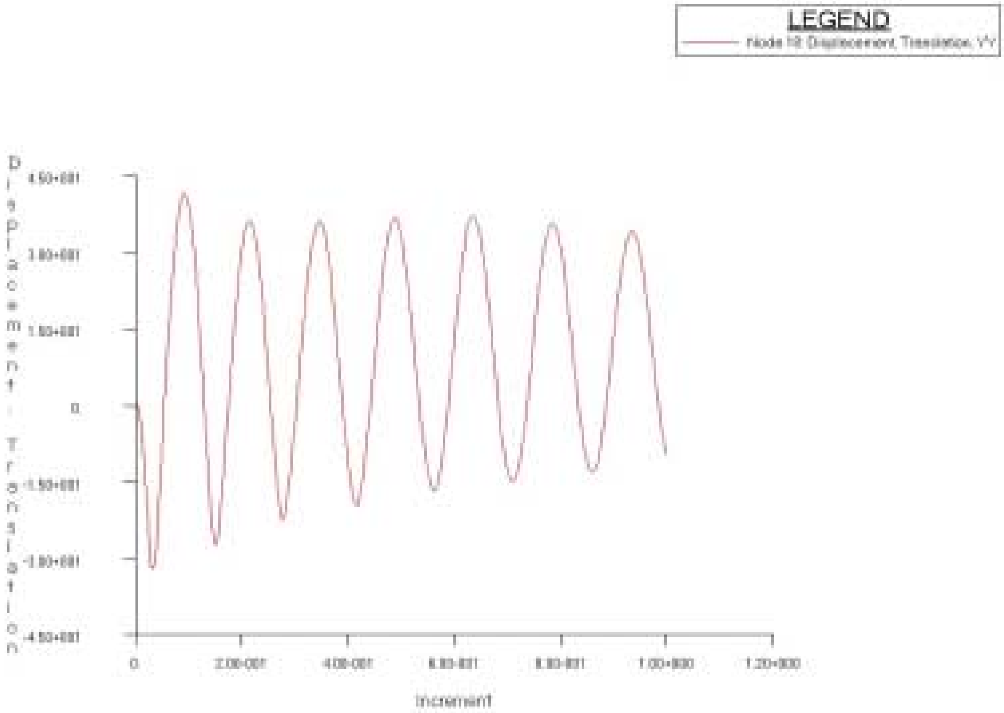
Select Y Result: **Displacement, Translation**

Quantity:	**Y Component**
X:	**Global Variable**
Global Variable:	**Time**

Apply

The Results are shown in **Figure 20.5** below. Note the initial deflection in the negative direction is cut short due to the contact.

Figure 20.5: Results including the contact.



17. Remove the contact from the current Load Case in the problem.

■ **Load Cases**

Action:	**Modify**
Select Load Case to Modify:	**Impulse_Load**
Assigned Loads/BCs:	**beam** **rigid_stop**

In the spreadsheet that appears, select the two rows that have type Contact.

Remove Selected Rows

OK

Apply

18. Reactivate the damping. Analyze this model damped. Turn the damping constitutive model on for the material *steel*.

◼ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Existing Materials:	**steel**

Cancel

(Close the **Input Properties** form that automatically appears.)

Change Material Status...

Inactive Constitutive Models:	Select to make active, **Damping**

Apply

Cancel

Apply

19. Modify the time dependent Field.

Modify the field we created earlier, changing the time history for a sudden loading that remains constant.

◼ **Fields**

Action:	**Modify**
Object:	**Non Spatial**
Method:	**Tabular Input**
Select Field to Modify:	**impulse**

Rename Field As: **constant**

Table 20.2: Fill in the table as shown below in the **Input Data** form.

Time (t)	Value
0.0	0.0
0.015	1.0
0.03	1.0
1.0	1.0

OK

Apply

Note: Although we modified the field and changed its name, it is still associated to the 6,000 lb load applied at the tip of the beam. We have simply modified its time dependent behavior.

20. Set up the model and submit the Analysis.

Modify the **Load Step** for contact Analysis.

◾ **Analysis**

Action: **Analyze**

Object: **Entire Model**

Method: **Full Run**

Available Jobs: **impulse**

Job Name: **sudden**

Load Step Creation...

Available Job Steps: **Impulse Step**

Solution Type: **Transient Dynamic**

Solution Parameters...

Linearity: **Linear**

Load Increment Parameters...

Increment Type: **Fixed**

Time Step Size: **0.005**

Total Time: **1.0**

OK

OK

Select Load Case...

Available Load Case: **Impulse_Load**

OK

Apply

Cancel

Load Step Selection...

Make sure only **Impulse Step** is selected.

OK

Apply

21. Read and plot Results with contact.

Repeat **Step 12**, reading in the results. Select the new Job created, and plot the displacement with respect to time for the right top node at the free end.

Action: **Read Results**

Object: **Result Entities**

Method: **Attach**

Available Jobs: **sudden**

Select Results File... **sudden.t16**

OK

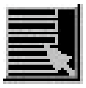
Apply

◆ **Results**

Action: **Create**

Object: **Graph**

Press the icon to select to select to the **Results Cases**. Node 18 should still be selected.



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

Clear

Filter Method: **String**

Filter String: ***A4***

Filter

Apply

Close

Y: **Result**

Select Y Result: **Displacement, Translation**

Quantity: **Y Component**

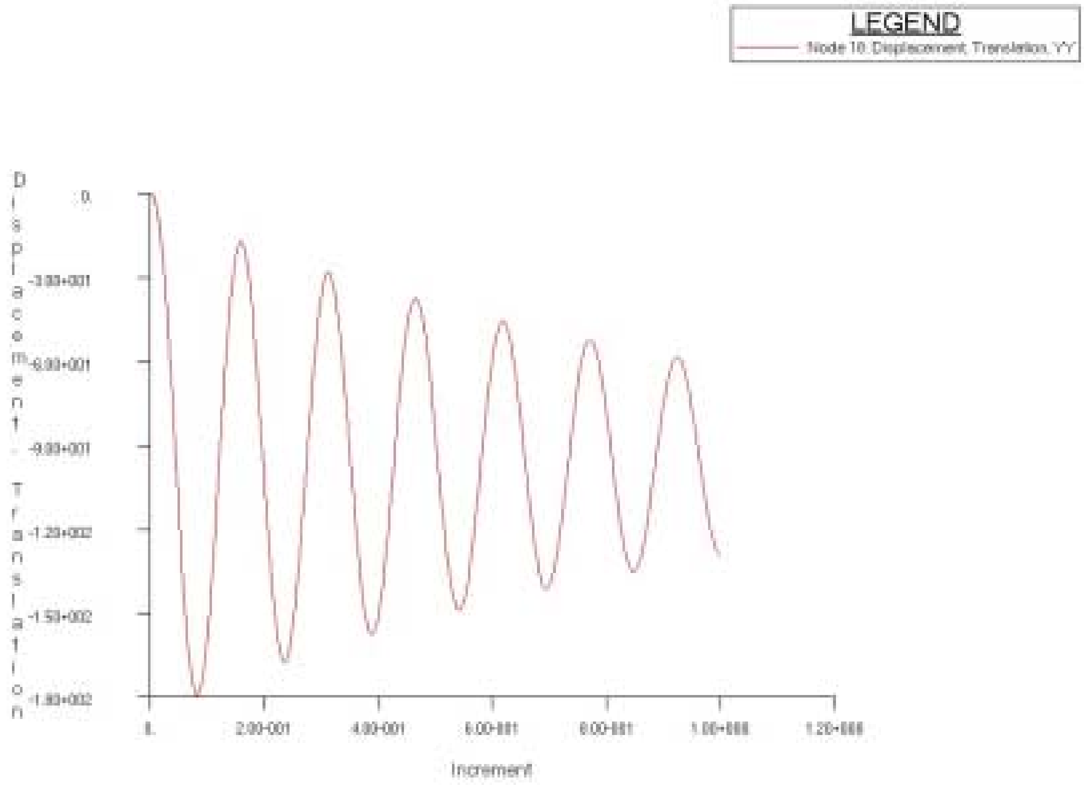
X: **Global Variable**

Global Variable: **Time**

Apply

The results are showing in **Figure 20.6** below. This Analysis was done to show that a sudden application of a load in dynamics results is roughly twice the deflection of the static Analysis. As the response attenuates, the deflection diminishes towards the static value.

Figure 20.6: Results with a sudden application of a load.



Close the database and quit PATRAN.

This concludes the exercise**.**