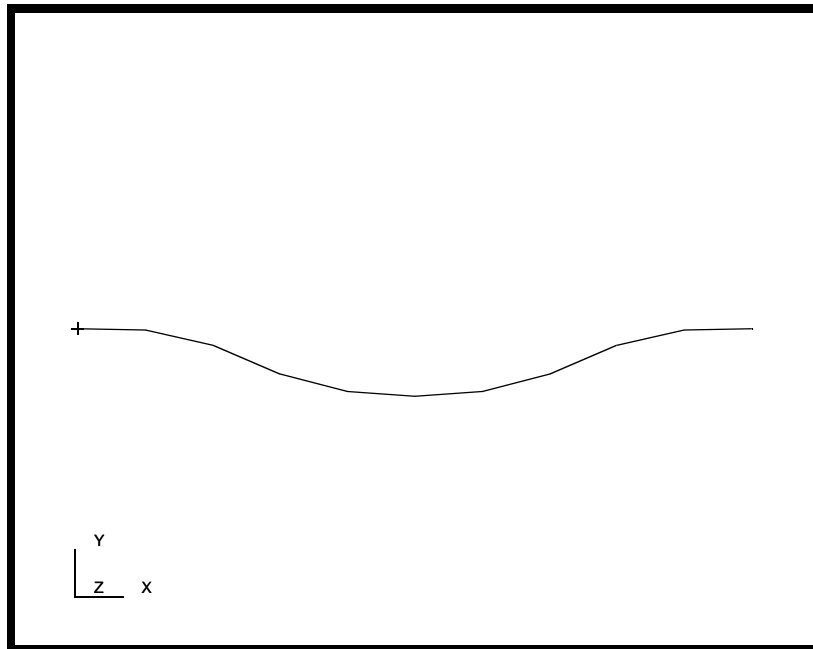

Lesson 5

Nonlinear Transient Analysis of Vibrating Wire



Objectives:

- Develop a 1D model that represents a wire of constant section clamped at both ends, and slowly loaded with a point force at the middle of the wire
- Create another load step in which the force is removed instantly and the wire is allowed to vibrate.
- Produce two animations of the deformation, one created with quick animation and the other showing true deformation (created using Insight)



Model Description:

The problem has the following physical properties:

Horizontal Wire's length	= 10cm
Circular Cross Section's radius	= 0.1 cm
Young's Modulus	= 1×10^2 N/cm ²
Poisson's ratio	= 0.3
Density	= 7.4×10^{-4} kg/cm ³
Force	= 1,000 N

Exercise Steps:

- The force is large enough to expect large displacements.
- The wire is modeled with beam elements.
- Create a straight curve, mesh with 10 1D Bar2 elements.
- Create material properties and element properties.
- Fix both ends of the wire and create the point load.
- Create a first, nonlinear static step using the default load case.
- Create a new load case and include only the fix end boundary condition - this case would have no load thus.
- Create a second, nonlinear transient step using the newly created load case.
- Select the steps in the correct order and run the analysis.
- Read the results and produce the requested animations.

Exercise Procedure:

1. Create a new database and name it **wire.db**.

File/New ...

New Database Name:

wire.db

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 pick the following options

Analysis Code:

MSC.Marc

OK

2. Create the geometry for the wire.

■ Geometry

Action:

Create

Object:

Curve

Method:

XYZ

Vector Coordinates List:

<10, 0, 0>

Origin Coordinates List:

[0, 0, 0]

Apply

Turn on the labels by using the following toolbar icon:



Show Labels

A horizontal line will appear in the viewport, representing the wire model.

3. Create the FEM mesh for the wire.

■ Elements

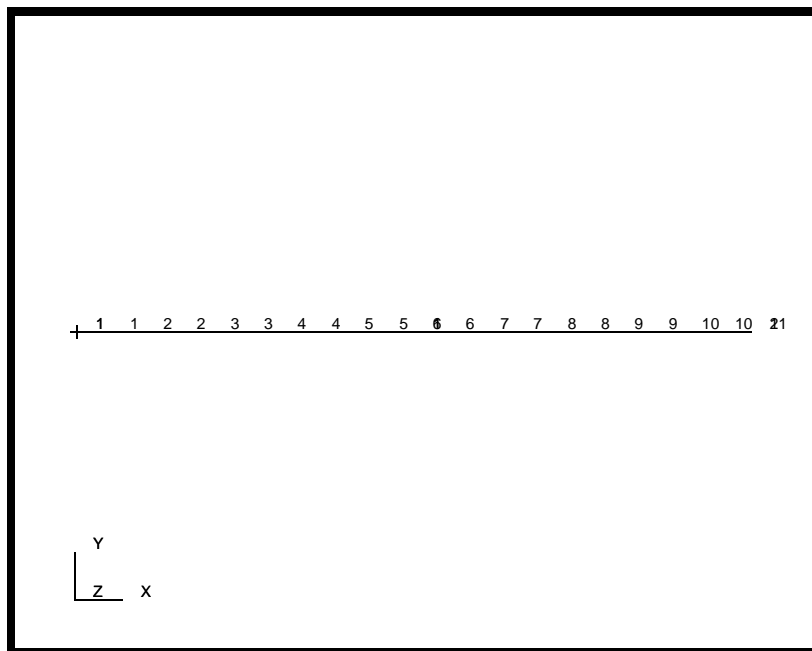
Action:

Create

<i>Object:</i>	<input type="text" value="Mesh"/>
<i>Type:</i>	<input type="text" value="Curve"/>
<i>Global Edge Length:</i>	<input type="text" value="1.0"/>
<i>Element Topology:</i>	<input type="text" value="Bar2"/>
<i>Curve List:</i>	<input type="text" value="select the wire"/>
<input type="button" value="Apply"/>	

Your model should appear like the one shown in Figure 5.1:

Figure 5.1 - Meshed Wire



4. Create a linear elastic isotropic material named **mat1** using the specified values for E , ν , and ρ .

■ Materials

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Isotropic"/>
<i>Method:</i>	<input type="text" value="Manual Input"/>
<i>Material Name:</i>	<input type="text" value="mat1"/>

Input Properties...

Elastic Modulus:

Poisson's Ratio:

Density:

OK

Apply

5. Create a 1D bar in the XY-plane element property named **wire** for the curve. Since you have meshed the curve with Bar2 elements, these elements will inherit this property.

■ Properties

Action:

Dimension:

Type:

Property Set Name:

Options:

Input Properties ...

Material Name:

Section Height:

Section Width:

OK

Select Members:

Add

Apply

6. Constrain all degrees of freedom on both ends of the wire.

■ Loads/BCs

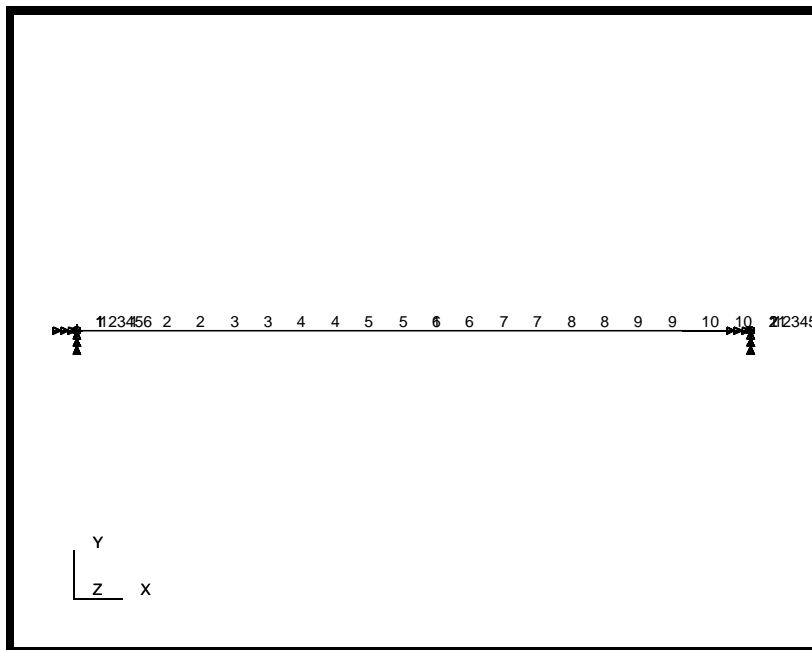
Action:

Object:

Type:	<input type="text" value="Nodal"/>
New Set Name:	<input type="text" value="clamps"/>
<input type="button" value="Input Data..."/>	
Translations $\langle T1, T2, T3 \rangle$:	<input type="text" value="< 0, 0, 0 >"/>
Rotations $\langle R1, R2, R3 \rangle$:	<input type="text" value="< 0, 0, 0 >"/>
<input type="button" value="OK"/>	
<input type="button" value="Select Application Region..."/>	
Select Geometric Entities:	<input type="text" value="select both end points (1 & 2)"/>
<input type="button" value="Add"/>	
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	

The boundary conditions will be displayed as shown in Figure 5.2:

Figure 5.2 - Model with Boundary Conditions applied



-
7. Create an applied force named **push**, consisting of 1000 Newtons in the negative y-direction.

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Force"/>
<i>Type:</i>	<input type="text" value="Nodal"/>
<i>New Set Name:</i>	<input type="text" value="push"/>
<input type="button" value="Input Data..."/>	
<i>Force <F1, F2, F3>:</i>	<input type="text" value="< 0, -1000, 0 >"/>
<input type="button" value="OK"/>	
<input type="button" value="Select Application Region..."/>	
<i>Geometry Filter:</i>	<input checked="" type="radio"/> FEM

In order to better select the center node of the mesh (where the load will be applied), first increase the node size using the following toolbar icon:



Node Size

<i>Select Nodes:</i>	<input type="text" value="select the middle node of the wire (Node 6)"/>
<input type="button" value="Add"/>	
<input type="button" value="OK"/>	
<input type="button" value="Apply"/>	

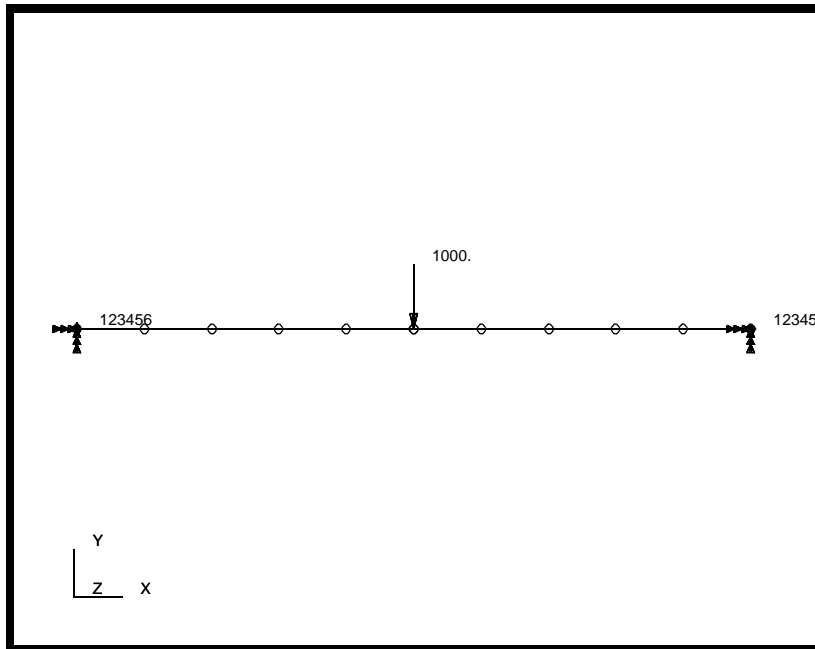
An arrow will appear on your screen in the center of the model, pointing downward. Turn off the labels now, using the following toolbar icon:



Hide Labels

Your viewport should now appear as shown in Figure 5.3:

Figure 5.3 - Model with Initial Force Applied



Shrink the node size back down by reselecting the following toolbar icon:



Node Size

8. Create a nonlinear static step named **push**.

■ Analysis

Action:

Analyze

Object:

Entire Model

Method:

Full Run

Job Name:

wire

Load Step Creation...

Job Step Name:

push

Solution Type:

Static

Output Requests...

Increments between Writing Results:

(Results will be written at the end of step regardless of this value, so your entering of “0” means that you will get results at the end of the step only).

Now add Result Type Velocity to your nodal results list.

(Leave other selections as set up by default)

9. Create a new load case named **unloaded**.

■ **Load Cases**

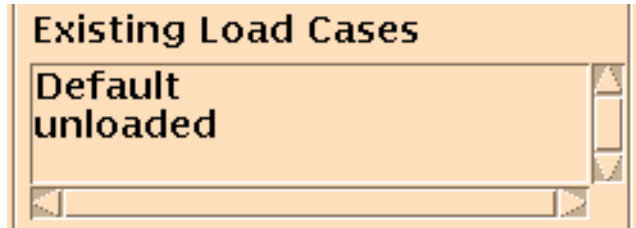
Action:

Load Case Name:

Select LBCs to Add to Spreadsheet:

Notice the force disappears from the viewport. This is because the newly created load case has been made current. Therefore, if you were to create new loads/BCs now - which is not the case - these would be automatically added to the new load case.

To quickly verify the setup of each load case, click on them in the *Existing Load Cases* panel.



As you click on each of them, you can see the contained loads in a spreadsheet format on the *Prioritize Loads/BCs* form. Clicking on the other load case changes the form seen on the screen, as you can see by the changing included loads as well as the changing name in the *Load Case Name* panel.

Select the load case **unloaded**, and close the *Prioritize Loads/BCs* panel.

OK

10. Create a nonlinear transient step named **vibrate**, finish setting up the analysis, and run it.

■ Analysis

Job Name:

wire

Load Step Creation...

Job Step Name:

vibrate

Solution Type:

Transient Dynamic

Select Load Case...

Available Load Cases:

unloaded

OK

Solution Parameters...

**Load Increment
Parameters...**

Increment Type:

Adaptive

Trial Time Step Size:

.05

Minimum Time Step:

.0001

Max # of Steps:

500

OK

OK

Output Requests...

Select Nodal Results...

(add) Velocity

OK

OK

Apply

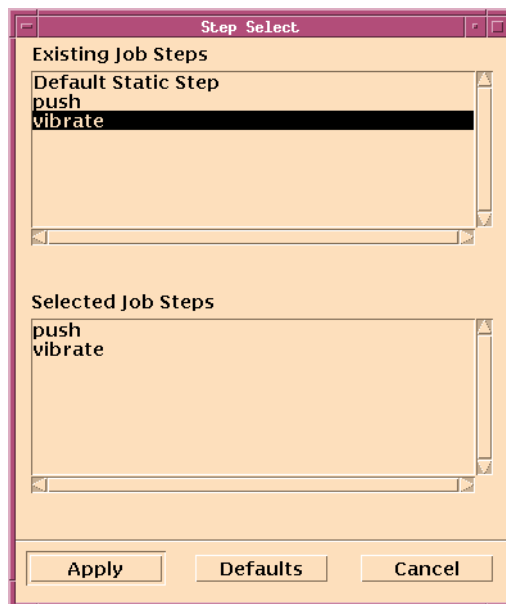
Cancel

Load Step Selection...

Selected Job Steps:

**push
vibrate**

Be sure to select push and vibrate in that exact order. You cannot remove the default step until you have selected at least one other step. When finished, the form should appear as follows:



OK

Apply

When the job is done there will be a results file titled **wire.t16** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at the **wire.log**, **wire.sts**, **wire.out** files to monitor the status. A more convenient method would be to use the **Monitor** action in the **Analysis** application.

Action:

Monitor

Object:

Job**View Status File...**

A job exit number of **3004** will verify if the job has completed successfully.

When the job finishes, viewing the **wire.log** file will include the last lines of the job's message file, which will let you read a Job Time Summary and the message "*successful completion to an MSC.Marc analysis.*"

- Once the job has finished, read in the results.

■ Analysis

Action:

Read Results

Object:

Result Entities

Method:

Attach**Select Results File...**

Selected Results File:

wire.t16**OK****Apply**

The viewport will disappear from view while the results are read in, and reappear as soon as the reading in has completed.

- Do an animation using the Results form.

■ Results

Action:

Object:

Method:

Click on the **View Subcases** icon then the **Select Subcases** to bring up the *Select Result Case* form



Select Result Case(s):

Filter Method

Select Result Case(s):

Select VectorResult:

■ **Animate**

Change the display attributes of the vector length.



Vector Definition Length

Screen Scaled

We do this because we intend to display the vector displacements scaled up to their actual magnitudes, with the largest magnitude shown as 0.1 the viewport size.

Click on the **Animation Options** icon to bring up that form



Number of Frames:

After some time involved in the preparation of the graphics' frames, these will be shown in rapid sequence. Notice the vectors are not aligned with the vertical, illustrating the need for a nonlinear analysis if accuracy is desired. This will be seen more clearly if you again click on Plot Type Options and make the following changes:



Select Vector Result:

Velocity, Translation

Apply

Notice how the velocity vectors of the wire do not point completely vertical, indicating that there are horizontal as well as vertical displacements of the node of the wire. However, the horizontal displacements are much smaller than vertical ones, and can be neglected.

Close the database and quit PATRAN.

This concludes this exercise.

