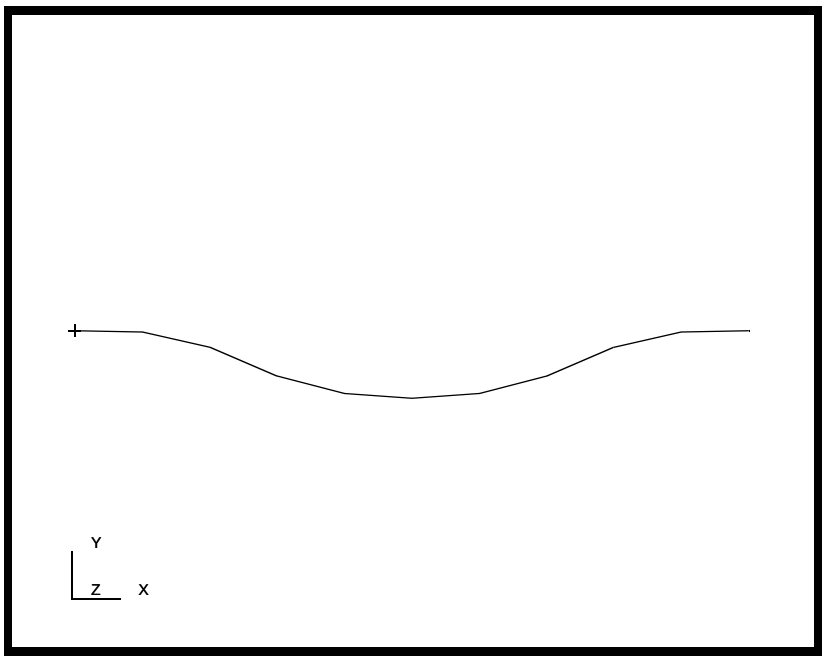
APPENDIX 1

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	= $1x10^2$ N/cm^2
Poisson's ratio	= 0.3
Density	= $7.4x10^{-4}$ 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 stes 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/ADVANCED_FEA**

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.

◆ **Finite Elements**

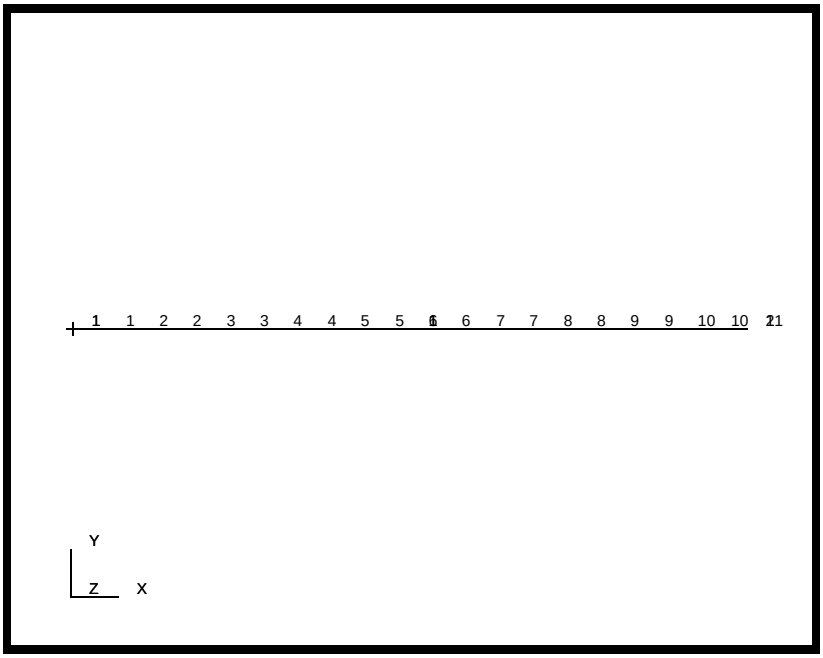
Action: **Create**

Object:	**Mesh**
Type:	**Curve**
Global Edge Length:	**1.0**
Element Topology:	**Bar2**
Curve List:	select the wire

Apply

Your model should appear like the one shown in Figure 14.1:

Figure 14.1 - Meshed Wire



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

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	**mat1**

Input Properties...	
Elastic Modulus:	**1e7**
Poisson's Ratio:	**0.30**
Density:	**7.4e-4**
Apply	
Cancel	

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:	**Create**
Dimension:	**1D**
Type:	**Beam in XY Plane**
Property Set Name:	**wire**
Options:	**Circular Section**
	Standard Formulation
Input Properties ...	
Material Name:	**mat1**
Section Radius (ave):	**0.1**
OK	
Select Members:	select the curve
Add	
Apply	

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

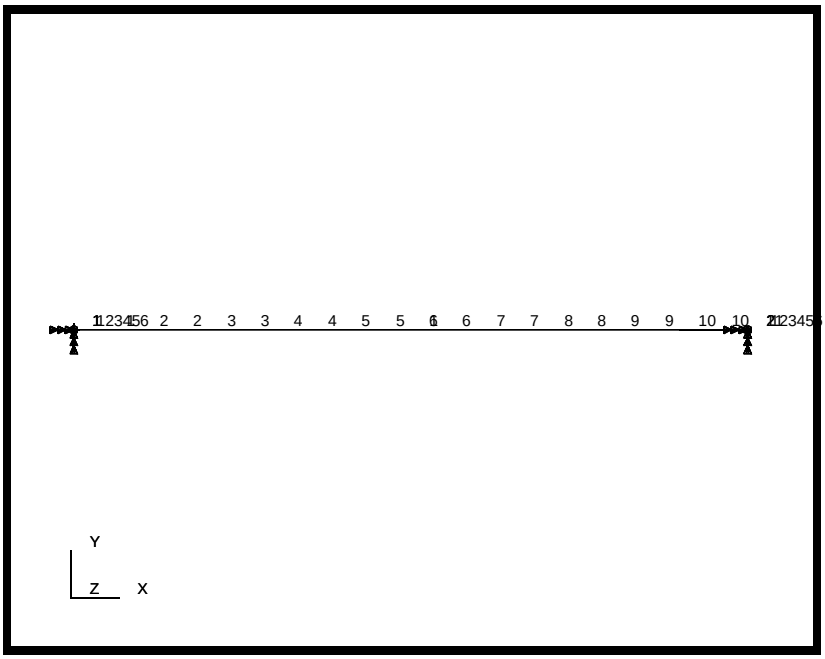
◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**

New Set Name:	**clamps**
Input Data...	
Translations <T1,T2,T3>:	**< 0, 0, 0 >**
Rotations <R1,R2,R3>:	**< 0, 0, 0 >**
OK	
Select Application Region...	
Select Geometric Entities:	select both end points (1 & 2)
Add	
OK	
Apply	

The boundary conditions will be displayed as shown in Figure 14.2:

Figure 14.2 - Model with Boundary Conditions applied



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

Action:	**Create**

Object: **Force**

Type: **Nodal**

New Set Name: **push**

Input Data...

Force <F1, F2, F3>: **< 0, -1000, 0 >**

OK

Select Application Region...

Geometry Filter: ◆ **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**

Select Nodes: select the middle node of the wire (Node 6)

Add

OK

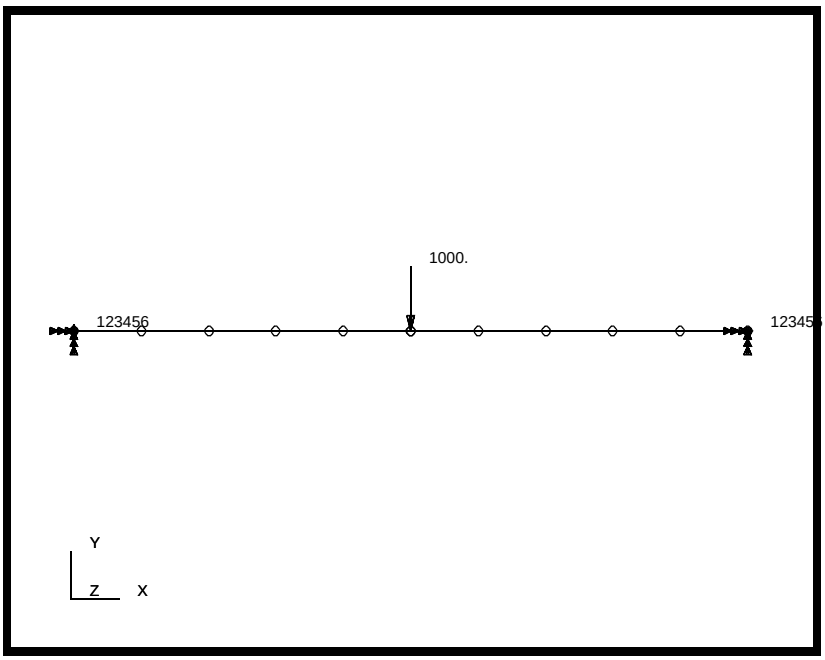
Apply

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 14.3:

Figure 14.3 - 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**
Step Creation...	
Job Step Name:	**push**
Solution Type:	**Nonlinear Static**
Output Requests...	
Increments Between Outputs:	**0**

(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)

Reaction Forces: **On**

(Leave other selections as set up by default)

OK

Apply

Cancel

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

◆Load Cases

Action: **Create**

Load Case Name: **unloaded**

Assign/Prioritize Loads/BCs

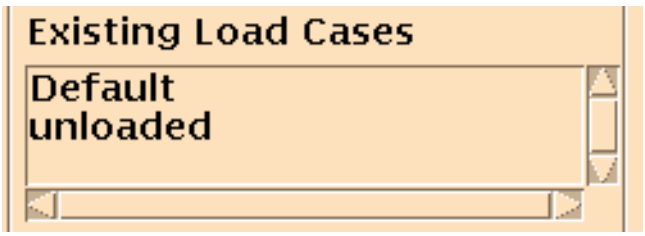
Select LBCs to Add to Spreadsheet: **Displ_clamps**

OK

Apply

Notice the force disappears from the viewport. This is becase 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**

Step Creation...

Job Step Name: **vibrate**

Solution Type: **Nonlinear Transient Dynamic**

Select Load Cases...

Available Load Cases: **unloaded**

OK

Solution Parameters...

Max No. of Increments Allowed: **500**

Delta T: **.0001**

Time Duration of Step: **.05**

Max Error in Mid Incremental Residual: **100**

OK

Apply

Cancel

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:

Apply

Apply

The analysis job will take (on average) about 5 minutes to run. When the job is done there will be a results file titled **wire.fil** in the same directory you started MSC/PATRAN in.

Again, you can monitor the progression of the job by looking at **wire.msg** and **wire.sta** with the ***more*** command. Also, you may use ***ps*** and ***tail -lf wire.sta*** to monitor the status.

If you were to select a Check Run in the analysis form, only the **[afea1]** part of the analysis will run. Likewise, if the model would have a fatal error as diagnosed by **[afea1]**, the **[afeamain]** part will not be executed. When the job has finished, the command ***ps*** will not return **[afea]**.

When the job finishes, entering ***tail wire.msg*** to see the last lines of the job's message file will let you read a Job Time Summary and the message "*Successful termination of afeacontrol.*"

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

◆ **Analysis**

Action:	**Read Results**
Object:	**Result Entities**
Method:	**Translate**
Select Results File...	
Selected Results File:	**wire.fil**
OK	
Apply	

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

12. Do an animation using the Results form.

◆ **Results**

Action:	**Create**
Object:	**Marker**
Method:	**Vector**

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


Select Result Case:	**unloaded, 500 Subcases**
Filter Method	**All**
Filter	
Apply	
Close	
Select VectorResult:	**Deformation,Displacement**

Change the display attributes of the vectot 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: **20**

Apply

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 Results: **Velocity,Translational**

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.