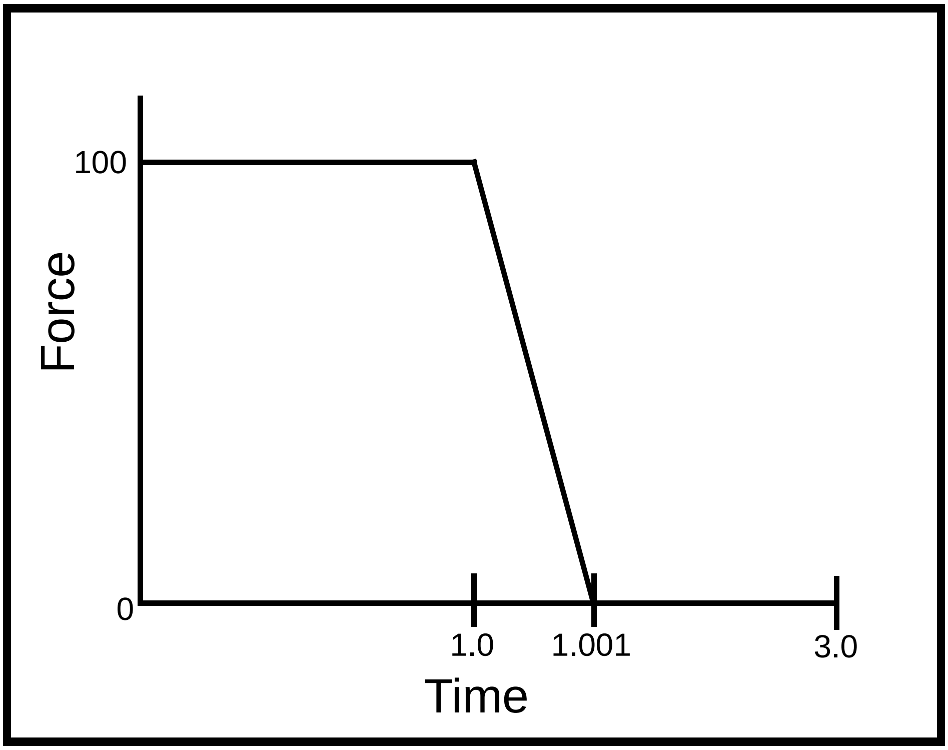
LESSON 3

Transient Response of a Rocket



Objectives:

- Develope a finite element model that represents an axial force (thrust) applied to a rocket over time.
- Perform a linear transient analysis of the model.
- Compare results to analytic calculations.

Exercise Description:

An axial force (thrust) is applied to a rocket over time. Using three elements to model the rocket as an unconstrained structure, determine the displacements of the base of the rocket with respect to time.

The rocket and applied thrust has the following properties:

Length = 140 inches

Area = 1.0 in^2

$\nu = 0.30$

$\rho = 0.1\ lb/in^3$

E = 1.0 E+4 lb/in^2

Force = 100 lbs

Time vs. Force History:

time (t)	Force(f)
0.0	**100.0**
1.0	**100.0**
1.001	**0.0**
3.0	**0.0**

Exercise Procedure:

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

File/New ...

New Database Name: **rocket.db**

OK

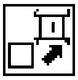
In the New Model Preference form set the *Analysis Code* to **MSC/ADVANCED_FEA.**

Analysis Code: **MSC/ADVANCED_FEA**

OK

2. Create the geometry for the rocket.

First, turn on entity labels using the following toolbar icon:

 **Show Labels**

◆ **Geometry**

Action: **Create**

Object: **Curve**

Method: **XYZ**

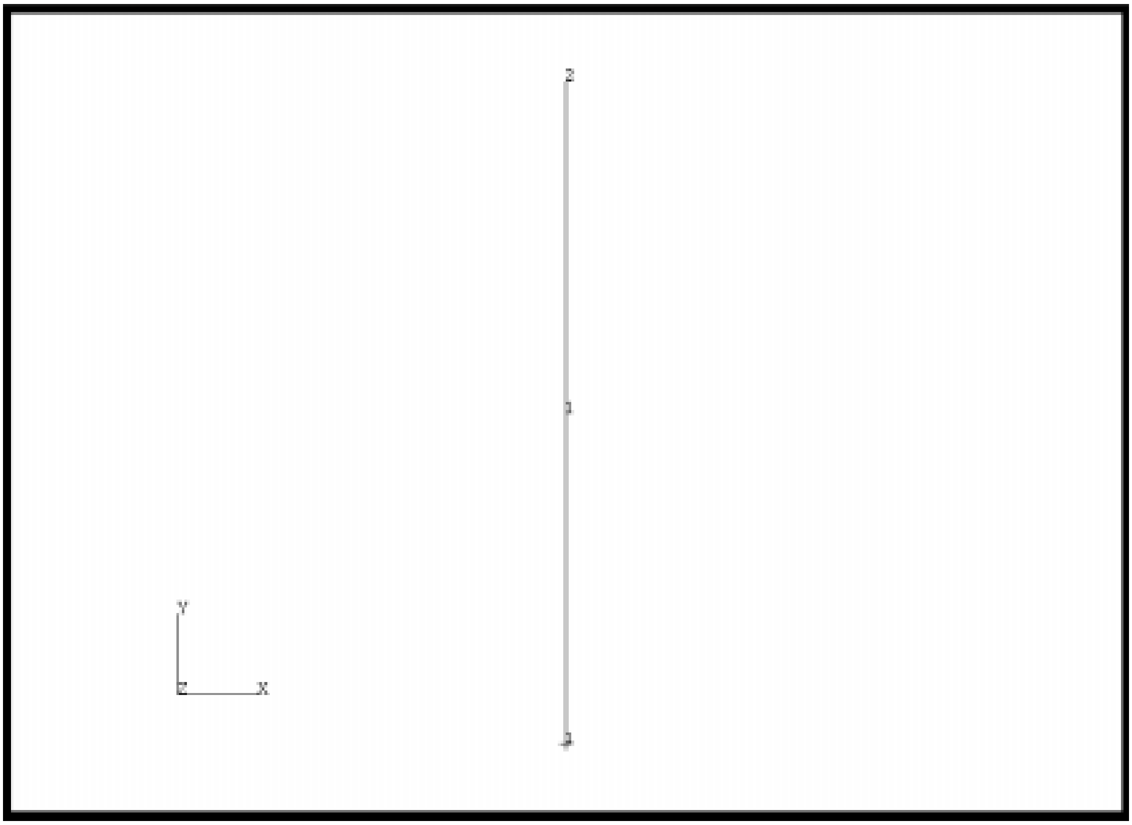
Vector Coord List: **< 0, 140, 0 >**

Origin Coord List: **[0, 0, 0]**

Apply

A line should appear in your viewport as shown in Figure 12.1:

Figure 12.1 - Line representing rocket



3. Create a mesh seed of 3 for the line.

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Type:	**Uniform**
Number:	**3**
Curve List:	**Curve 1**

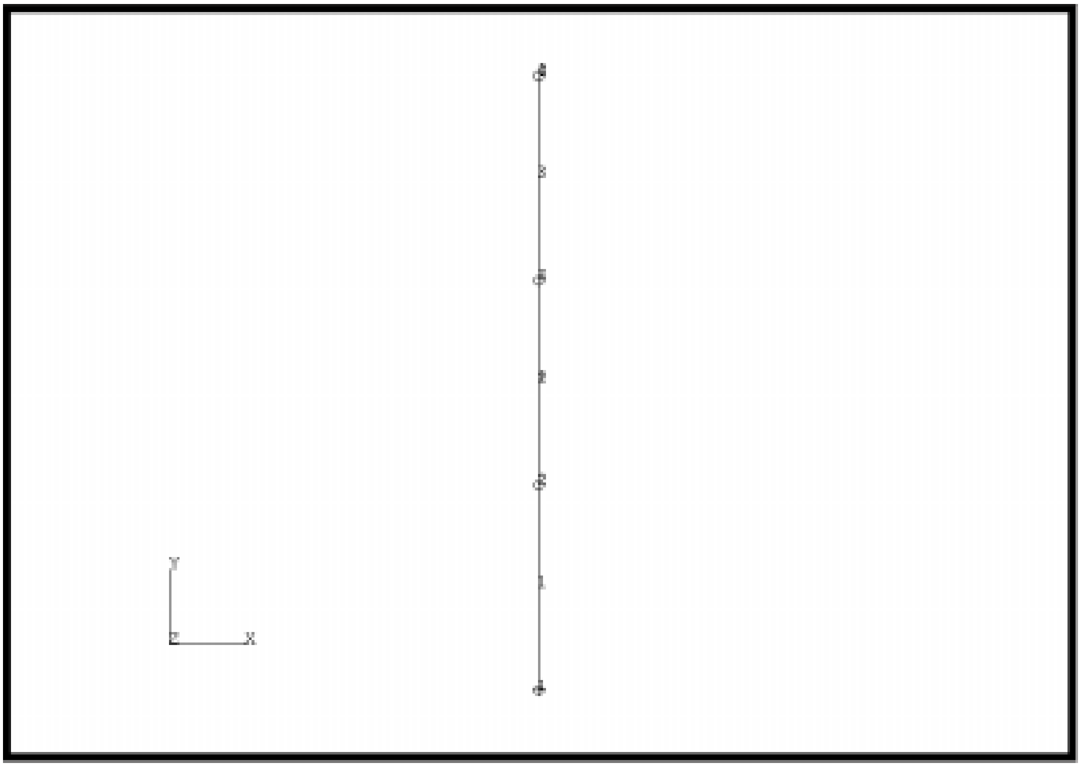
4. Now mesh the curve.

Action:	**Create**
Object:	**Mesh**
Type:	**Curve**
Curve List:	**Curve 1**

Apply

Your model should look like the one shown in Figure 12.2:

Figure 12.2 - Three element mesh of rocket



5. Next create a linear elastic isotropic material named **panel** using the specified values for E, υ, ρ.

◆ **Materials**

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

Input Properties...

Elastic Modulus:	1.0E4
Poisson's Ratio:	0.30
Density:	0.1

Apply

Cancel

6. Create a 1D bar in space element property named **bar**.

◆ **Properties**

Action:	**Create**
Dimension:	**1D**
Type:	**Beam in Space**
Property Set Name:	**bar**
Options:	**Circular Section**
	Standard Formulation

Input Properties...

Material Name:	**panel**
Section Radius:	**.5641897**
Definition of XY Plane	**< 0, 0, 1 >**

OK

Select Members:	**Curve 1**

Add

Apply

7. Create a Non Spatial Field named **time_history** with time as the active independent variable. Use the time history table given below to create the time vs. force field.

◆ **Fields**

Action:	**Create**
Object:	**Non Spatial**
Method:	**Tabular Input**
Field Name:	**time_history**
Active Independent Variable:	**Time**

Input Data...

Click on the corresponding box in the table and enter the values given in Table 1 into the Input Scalar Data box. Hit return and the number should appear in the table. Repeat this until all data values have been entered, then click

OK

Apply

Table 1: Force vs. Time History

time (t)	Force(f)
0.0	**100.0**
1.0	**100.0**
1.001	**0.0**
3.0	**0.0**

8. Create a time dependent loadcase named **time_vs_force**.

◆ **Load Cases**

Action: **Create**

Load Case Name: **time_vs_force**

Load Case Type: **Time Dependent**

Apply

9. Create an applied force named **thrust** with a force defined as **<0, 1, 0>** and a time dependence defined by the **time_history** field.

◆ **Loads/BCs**

Action: **Create**

Object: **Force**

Type: **Nodal**

New Set Name: **thrust**

Input Data...

Force <F1 F2 F3>: **< 0, 1, 0 >**

Time Dependence: **time_history**

OK

Select Application Region...

Geometry Filter: ◆ **FEM**

Select Nodes: **Node 1**

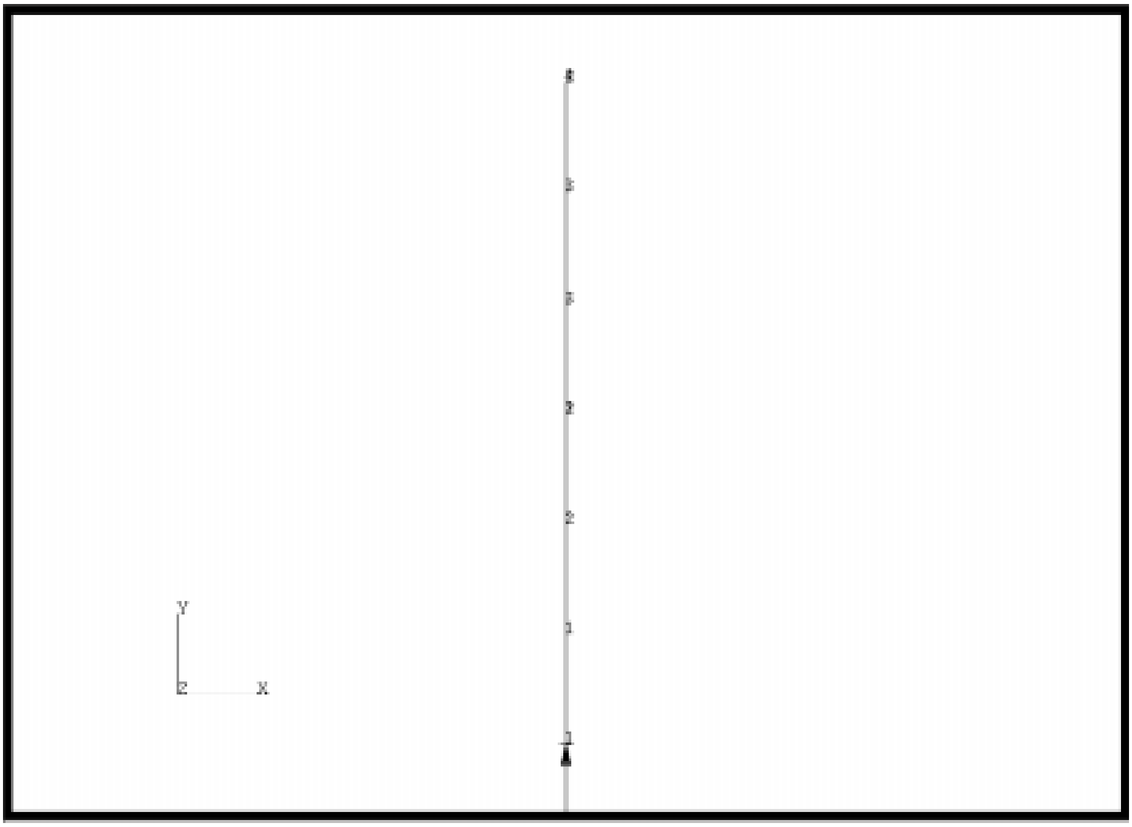
Add

OK

Apply

An arrow will appear on your screen as shown at the bottom of Figure 12.3:

Figure 12.3 - Applied "thrust" of rocket



10. Constrain all degrees of freedom except the Y direction on the line.

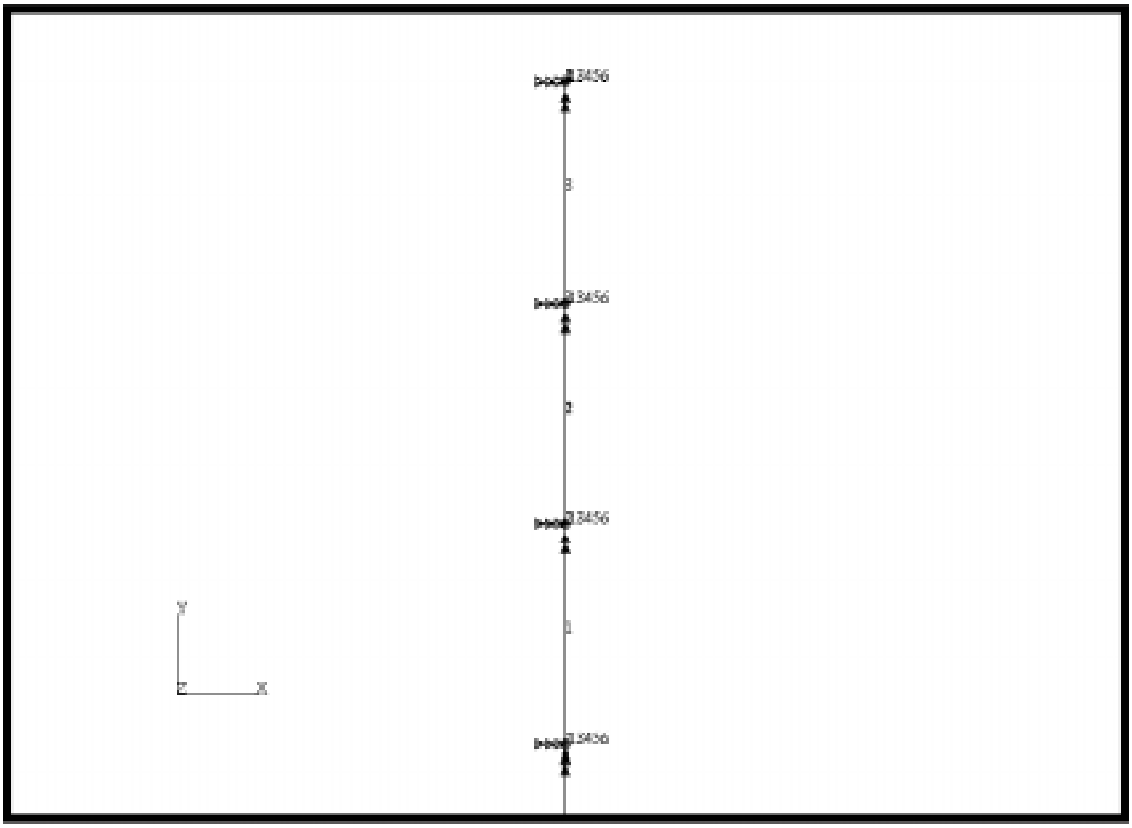
◆ **Load/BCs**

Action: **Create**

Object:	**Displacement**
Method:	**Nodal**
New Set Name:	**constraint**
Input Data...	
Translation <T1 T2 T3>:	**<0, , 0>**
Rotational <R1 R2 R3>:	**<0, 0, 0>**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**
Select Nodes:	**Node 1:4**
Add	
OK	
Apply	

Your screen will look like Figure 12.4:

Figure 12.4 - Rocket with applied boundary conditions



11. Create an analysis step named **take_off** using Step Creation. Then, select this new step and unselect the default static step under *Step Selection*.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Step Creation...	
Job Step Name:	take_off
Solution Type:	**Direct Linear Transient**
Solution Parameters...	
Delta-T:	0.05
Time Duration of Step:	3.0

OK

Select Load Cases...

Click on **time_vs_force** then click:

OK

Apply

Cancel

Step Selection...

Selected Job Steps: **take_off**

Apply

Apply

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

◆ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Translate**

Select Results File... **rocket.fil**

OK

Apply

13. To use XY-Plot change to the **Results** form.

◆ **Results**

Action: **Create**

Object: **Graph**

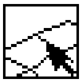
Method: **Y vs X**

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

Select Result Case:	**Time_vs_Force, 60 Subcases**
Filter Method	**All**
Filter	
Apply	
Close	
Y:	**Result**
Select Y Result:	**Deformation,Displacement**
Quantity:	**Y Component**
X:	**Global Variable**
Variable:	**Time**

Select the **Target Entity** icon



Target Entity:	**Nodes**
Select Nodes	**Node 1**
Apply	

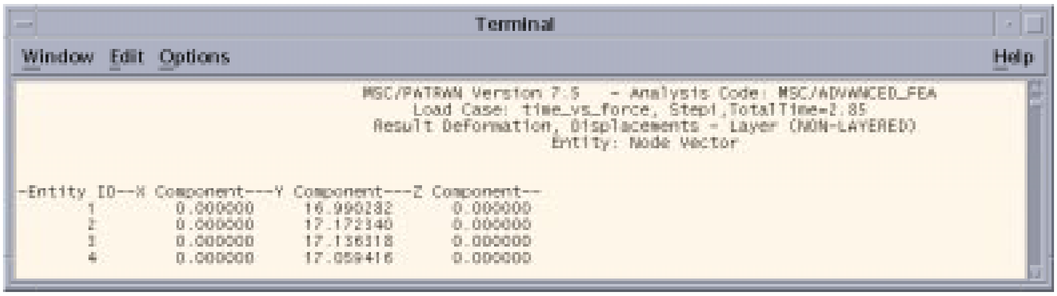
14. To obtain a **Text Report** change the *Object* to **Report** in the Results form

Action:	**Create**
Object:	**Report**
Method:	**Preview**

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


Select Result Case:	**Time_vs_Force, 60 Subcases**
Filter Method	**All**
Filter	
Apply	
Close	
Select Report Result:	**Deformation, Displacement**
Apply	

Click on **OK** if a warning appears for results only appearing in the analysis system. The Text Report appears in the unix window and looks like this:



Compare these results with the theoretical values.

Results Summary:

The displacements at node 1 can be compared to the analytical predictions given by Theory of Matrix Structural Analysis, J.S. Przemieniecki, McGraw-Hill, 1968, pg 367.

Time	Analytic Solution	P3/AFEA	% Diff
2.00	**10.8997**		
2.15	**11.7323**		

Close the database and quit PATRAN.

This concludes this exercise.

Time	Analytic Solution	P3/AFEA	% Diff
2.00	**10.8997**	**11.07**	**1.68**
2.15	**11.7323**	**12.20**	**3.98**