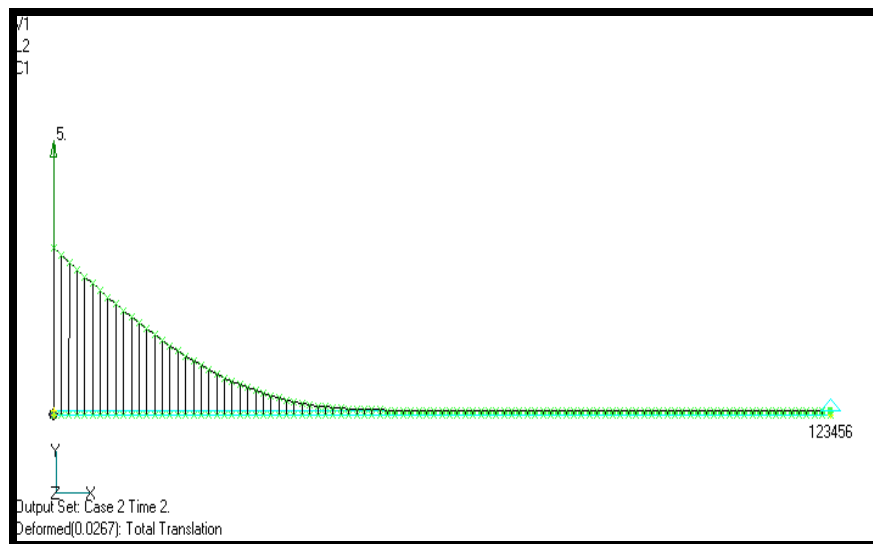


## WORKSHOP 11a

# *Lifting Bar from Ground with Gap Elements*



### Objectives:

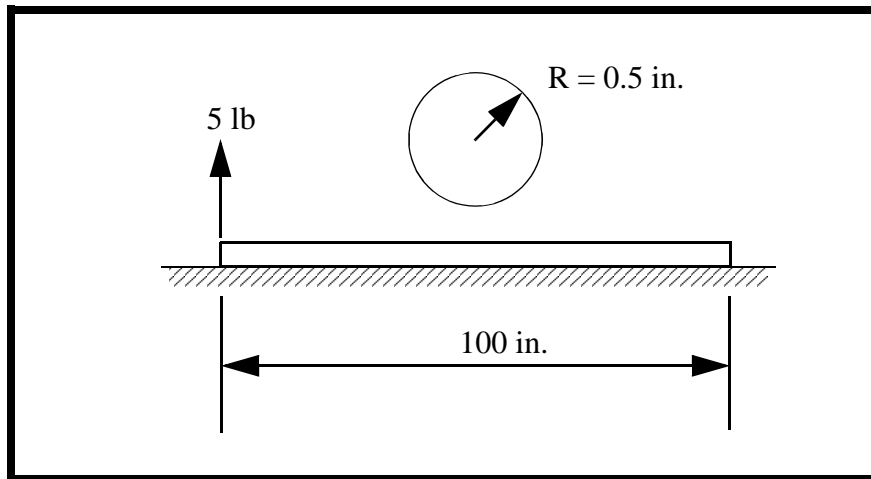
- Create a model of a beam in contact with the ground.
- Constrain the model.
- Load the model with its weight and a lifting force at one end.
- Determine the deflection of the beam and the length of the beam that is lifted from the ground.



### Model Description:

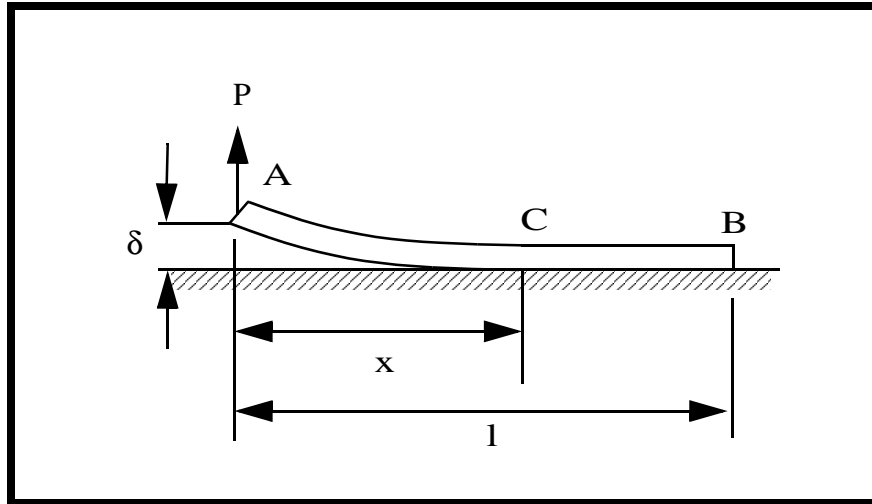
In this example, we will simulate a 100 in. long circular bar with a 1 in. diameter in contact with the ground under its own weight. A 5 lb vertical load will be applied to one end of the bar to lift it out of contact with the ground. We will determine the vertical deflection of the bar and the length that separates from the ground. We will also plot the shear and bending moment distribution in the bar.

**Figure 11a.1**



From “Strength of Materials, Part II: Advanced Theory and Problems”, by S. Timoshenko, Special Problems in Bending of Beams:

**Figure 11a.2**



for  $P < ql/2$

where the weight of the bar is  $w = ql$ ,

$$x = \frac{2P}{q}$$

$$\delta = \frac{qx^4}{24EI}$$

for our problem,

$$E = 29 \times 10^6$$

$$\rho = 7.33145 \times 10^{-4} \frac{lb \cdot sec^2}{in^4}$$

$$A = 0.785067 in^2$$

$$I = 0.0490874 in^4$$

$$g = 386.1 \frac{in}{sec^2}$$

$$q = \rho Ag = 0.0222227 \frac{lb}{in}$$

$$x = 45$$

$$\delta = 0.02667$$

## Exercise Procedure:

1. Start up MSC.Nastran for Windows V4.0.2 and begin to create a new model.

Double click on the icon labeled MSC.Nastran for Windows V4.0.2.

On the *Open Model File* form, select **New Model**.

*Open Model File:*

**New Model**

2. Create a material.

From the pulldown menu, select **Model/Material**.

**Model/Material...**

**Load...**

*Library Entry:*

**AISI 4340 Steel**

**OK**

**OK**

**Cancel**

3. Create a property called **beam**.

From the pulldown menu, select **Model/Property**.

**Model/Property...**

*Title:*

**beam**

**Elem/Property Type...**

*Line Elements:*

**Beam**

**OK**

*Material:*

**AISI 4340 Steel**

**Shape...**

*Shape:*

**Circular Bar**

*Radius:*

**0.5**

---

Orientation Direction (y):  Up

Create a property called **gap**.

Title:

Line Elements:

Gap

Compression Stiffness:

4. Create the geometry for the bottom curve to represent the ground.

**Geometry/Curve - Line/Project Points...**

X:

Y:

Z:

Create the geometry for the top curve to represent the beam.

**Geometry/Curve - Line/Project Points...**

X:

Y:

Z:

Turn off the workplane ruler.

**F2**

**Draw Workplane**

**Done**

**View/Autoscale (ctrl + A)**

- 5. Mesh the top curve with beam elements. Apply permanent constraints so that only motion in the xy plane is permitted.

**Mesh/Geometry/Curve...**

Select the *top* curve

**OK**

Property:

**1..beam**

**Node Param...**

Permanent Constraints:

TX    TY    TZ  
 RX    RY    RZ

**OK**

**OK**

**Method^**

**Global Axis**

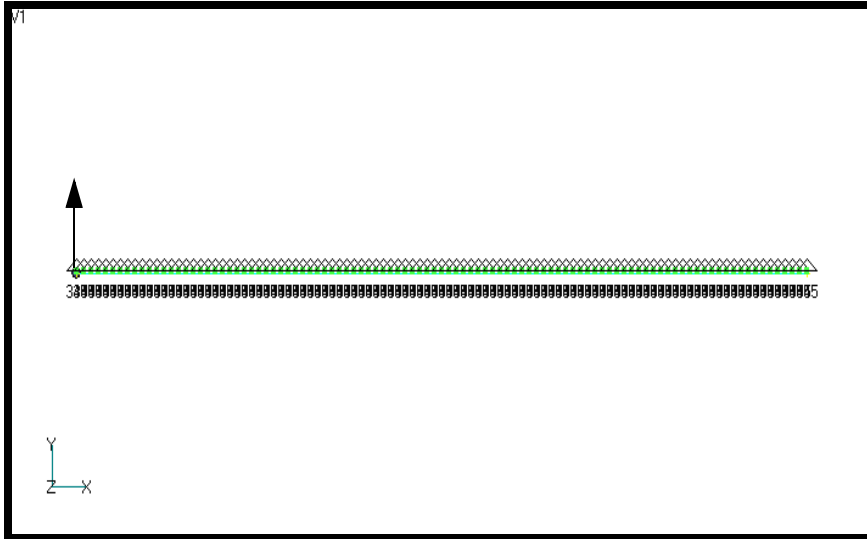
Direction:

● **Y Axis**

**OK**

Your model should look like the following figure.

Figure 11a.3



6. Mesh the bottom curve with nodes to represent the ground.

**Mesh/Geometry/Curve...**

Select the *bottom* curve

**OK**

**Node Param...**

Permanent Constraints:

TX    TY    TZ  
 RX    RY    RZ

**OK**

Generate:

Nodes

**OK**

7. Create the Gap Elements.

**Mesh/Connection/Closest Link..**

Select Nodes to Generate From.

**Method^**

**on Curve**

Select the *bottom* curve

**OK**

Select Nodes to Generate To.

<b>Method^</b>	<b>on Curve</b>
<i>Select the <b>top</b> curve</i>	
<b>OK</b>	
<i>Property:</i>	<b>2..gap</b>
<b>OK</b>	
<i>Direction:</i>	<input checked="" type="radio"/> <b>X Axis</b>
<b>OK</b>	

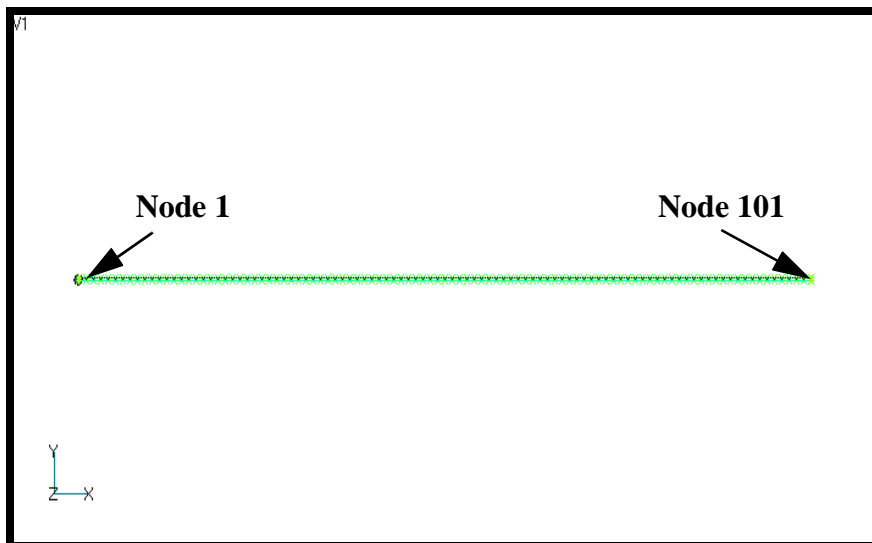
8. To view a more simplistic model, we will turn off the Node Permanent Constraints.

**View/Options...**

<b>Quick Options...</b>	
<i>Draw:</i>	<input type="checkbox"/> <b>Node - Perm Constraint</b>
<b>Done</b>	
<b>OK</b>	

Your model should look like the following figure.

**Figure 11a.4**



- 
9. Create the constraints for the model.

Before creating the appropriate constraints, a constraint set needs to be created. Do so by performing the following:

**Model/Constraint/Set...**

*Title:*

**constraint**

**OK**

Now define the relevant constraint for the model.

**Model/Constraint/Nodal...**

Select **Node 101**.

**OK**

Click on the **Fixed** button.

**Fixed**

**OK**

**Cancel**

Two load sets will be created for this analysis. The first will be the body load to represent the weight of the bar. This is used by itself to “seat” the gaps. The second will be the body load plus the lifting forces at the tip of the bar.

10. Create the first load.

Like the constraints, a load set must first be created before creating the appropriate model loading.

**Model/Load/Set...**

*Title:*

**grav**

**OK**

Create a body load. This represents the weight of the bar acting in the -y direction.

**Model/Load/Body...**

Check the top most **Active** box.

*Active*      Acceleration/Translation/Gravity:

X:       Y:       Z:

Select the Load Set Options for the Nonlinear Analysis.

**Model/Load/Nonlinear Analysis...**

*Solution Type:*      ● **Static**

11. Create the second load set by making a copy of the first load set.

**Model/Load/Copy...**

Rename the new load set

**Model/Load/Set...**

*Highlight 2..grav*

*Title:*     

Now apply a 5 lb force in the +y direction at the tip of the bar.

**Model/Load/Nodal...**

*Select Node 1*

*Highlight Force*

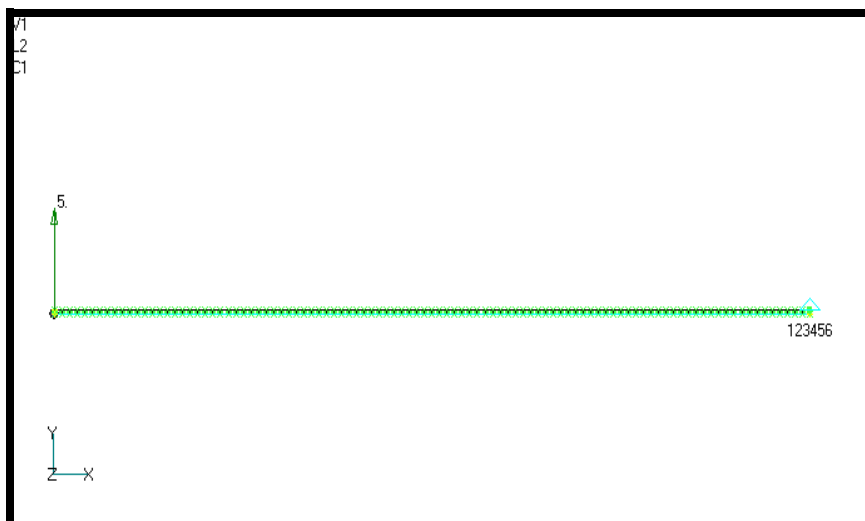
*FY*

**OK**

**Cancel**

Your model should look like the following figure.

**Figure 11a.5**



12. Create the input file and run the analysis.

**File/Export/Analysis Model...**

Type:

**10..Nonlinear Static**

**OK**

Change the directory to **C:\temp**.

File Name:

**prob11a**

**Write**

Additional Info:

**Run Analysis**

**Advanced...**

Problem ID:

**Bar Lift Gap**

**OK**

Output Requests:

**0..Print Only**

SUBCASE ID:

**1**

Loads=

**1..grav**

**Write Case...**

**OK**

*Loads=*

**2..grav+lift**

**OK**

**OK**

**OK**

When asked if you wish to save the model, click **Yes**.

**Yes**

*File Name:*

**prob11a**

**Save**

When the MSC.Nastran manager is through running, MSC.Nastran for Windows will be restored on your screen. When asked “OK to Read Nonlinear stresses and strains?” answer **Yes**.

**Yes**

The *Message Review* form will appear. To read the messages, you could select **Show Details**. Since the analysis ran smoothly, we will not bother with the details this time.

**Continue**

13. View the deflection results.

**View/Select...**

*Deformed Style:*

**Deform**

**Deformed and Contour Data...**

*Output Set:*

**2..Case 2 Time 2**

*Output Vectors/Deformation:*

**1..Total Translation**

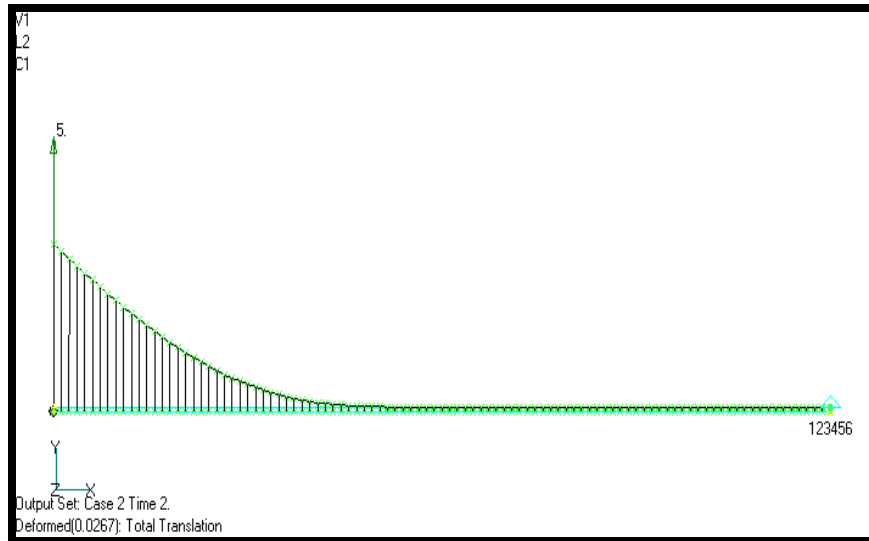
**OK**

**OK**

---

Your model should look like the following figure.

**Figure 11a.6**

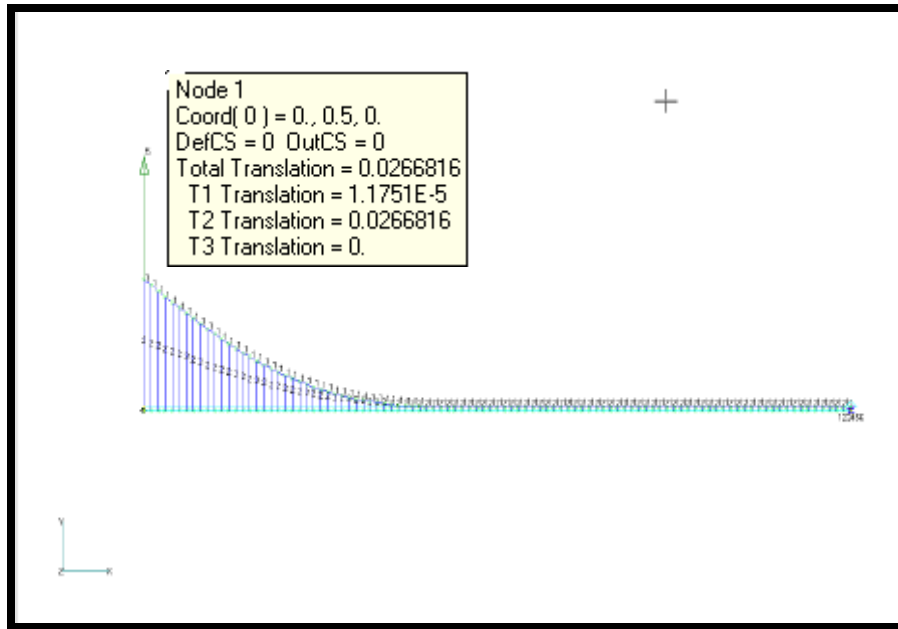


Set the Dynamic Query tool to Node. Find the tip deflection and the location of the first node still in contact with the ground.

- n Tip y deflection \_\_\_\_\_
- n x location of contact node \_\_\_\_\_

Your result should look like the following figure.

**Figure 11a.7**



14. View the beam shear and bending moment.

**View/Select...**

*Deformed Style:*  None

*Contour Style:*  **Beam Diagram**

**Deformed and Contour Data...**

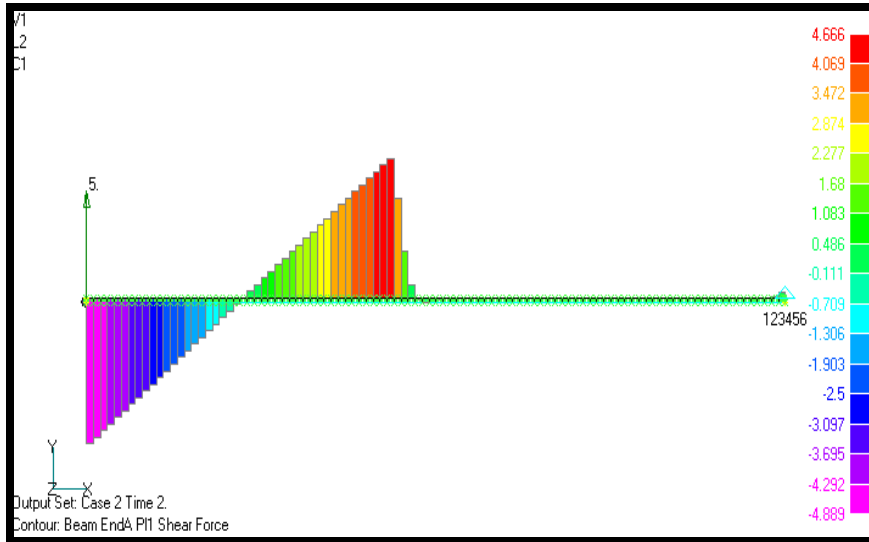
*Output Vectors/Contour:* **3018..Beam EndA P11 Shear Force**

**OK**

**OK**

Your model should look like the following figure.

**Figure 11a.8**



**View/Select...**

**Deformed and Contour Data...**

*Output Vectors/Contour:*

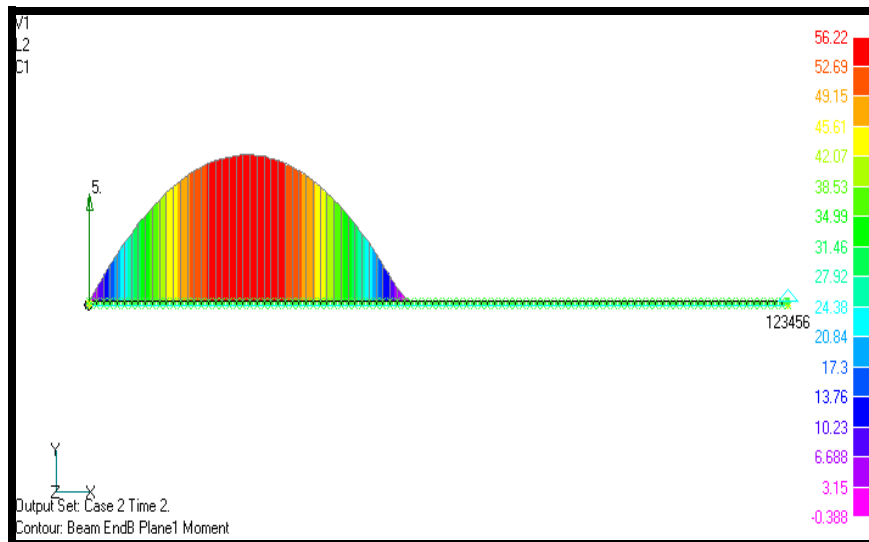
**3016..Beam EndB Plane1 Moment**

**OK**

**OK**

Your model should look like the following figure.

**Figure 11a.9**



T2 for Node 1 = 0.026665