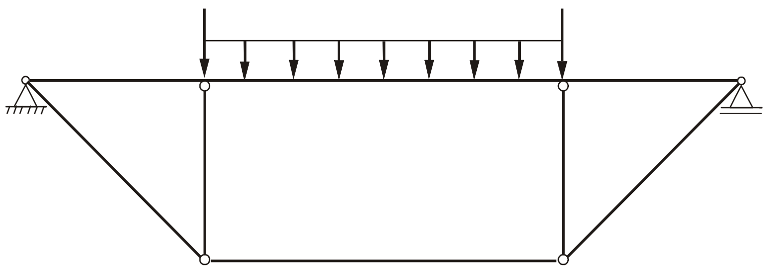
Workshop 2

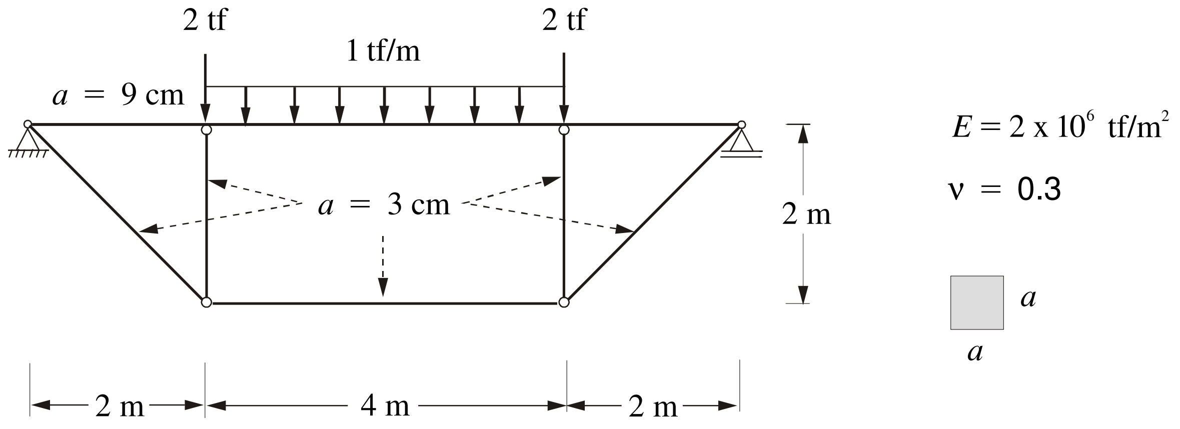
Linear Static Analysis of a Plane Frame



Objectives

- Create a finite element model by explicitly defining node locations and element connectivities.
- Define a MSC/NASTRAN analysis model comprised of bar elements.
- Run a MSC/NASTRAN linear static analysis.
- View analysis results.

Model Description



Consider the plane frame above, subjected to point and distributed loads. Define an analysis model comprised of bar elements.

Exercise Procedure

1. Start up **MSC/NASTRAN for Windows 4.5** and begin to create a new model.

   Double click on the icon for the **MSC/NASTRAN for Windows V4.5**.

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

   Turn off the workplane:

   Tools / Workplane (or **F2**) / ☐ **Draw Workplane / Done**

   View / Regenerate (or **Ctrl G**).

2. Create a material called **mat_1**.

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

Title	**mat_1**
Young's Modulus	**2e6**
Poisson's Ratio	**0.3**

   Select **OK / Cancel**.

   NOTE: In the *Messages Window* at the bottom of the screen, you should see a verification that the material was created. You can check here throughout the exercise to both verify the completion of operations and to find an explanation for errors which might occur.

3. Create properties called **prop_1** and **prop_2**.

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

Title	**prop_1**
Material	**mat_1**
	Elem / Property Type

Change the property type from **Plate** element (default) to **Bar** element.

Line Elements	**Bar**

Select **OK**.

To select the cross-sectional shape, click on **Shape**:

Shape	**Rectangular Bar**
H	**0.09**
Width	**0.09**
Orientation Direction (y)	✓ **Up**

Select **OK** / **OK**.

Title	**prop_2**

The material is already set to **mat_1** and the property type to **Bar** element.

To select the cross-sectional shape, click on **Shape**:

Shape	**Rectangular Bar**
H	**0.03**
Width	**0.03**
Orientation Direction (y)	✓ **Up**

Select **OK** / **OK** / **Cancel**.

4. Create the nodes for the frame model.

Create the first node of the model by selecting **Model** / **Node** (or **Ctrl N**).

X:	*Y*:	*Z*:
0	**2**	**0**

Select **OK**.

Repeat the process for the other 5 nodes:

Node	X	Y	Z	Select
2	**2**	**2**	**0**	**OK**
3	**6**	**2**	**0**	**OK**
4	**8**	**2**	**0**	**OK**
5	**2**	**0**	**0**	**OK**
6	**6**	**0**	**0**	**OK**

Select **Cancel**.

To fit the display onto the screen, select **View / Autoscale / Visible** (or **Ctrl A**).

5. Create the elements for the frame model.

First, display the node numbers:

View / Options / Quick Options (or **Ctrl Q**) / **Labels On / Done / OK**.

Choose **Model / Element** (or **Ctrl E**)

Property **prop_1**

When selecting the nodes, you may (if you wish) manually type in the endpoint nodes of the bar elements. However, it is easier to use the graphic interface and select the nodes on the screen using the mouse. Click in the first *Nodes* box and then select the nodes on the screen in the following order.

NOTE: The node nearest to the cursor is highlighted by a large yellow X - you don't have to click precisely on the node!

Nodes **1** **2**

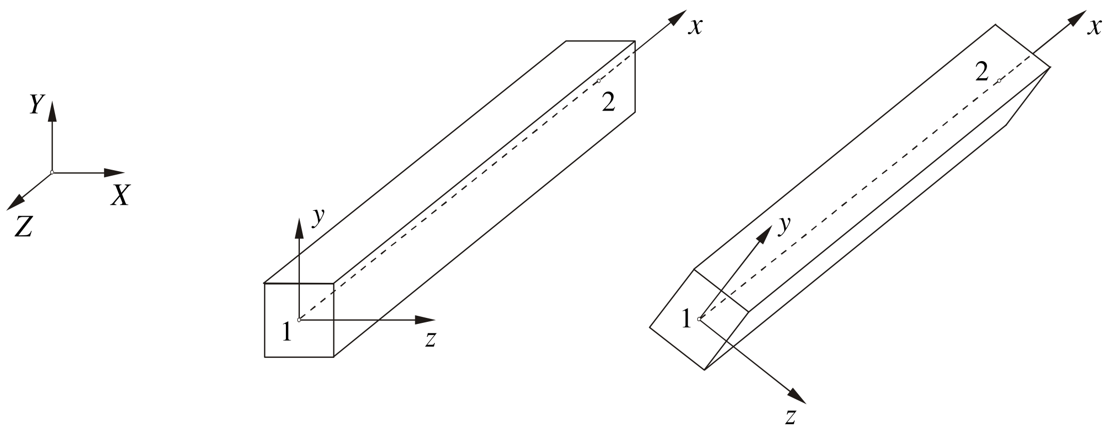
Now, specify a vector $\vec{v}$ to orient the element coordinate system xyz in the global XYZ space.

Select **Vector**.

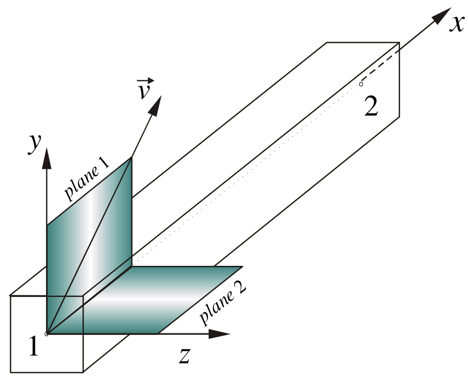
	X:	*Y*:	*Z*:
Base	***0***	***0***	***0***
Tip	***1***	***2***	***0***

Select **OK** / **OK**.

NOTE: One of the most difficult aspects of the Bar (or Beam) element for the first-time users is the understanding of the need to define an orientation vector. The best way to see the need is shown in the following figure, which indicates two out of an infinite number of possible orientations for the element connecting nodes 1 and 2.



The cross-sectional properties are defined in the element xyz system. The same element connecting nodes 1 and 2 may have, under bending, different contribution to the overall structural stiffness depending on how the cross section (or xyz system) is oriented in the global XYZ space. The orientation vector $\vec{v}$ will tell MSC/NASTRAN which one to choose. The element x-axis is defined by the coordinates of nodes 1 and 2 (it points from 1 to 2). The element y-axis is perpendicular to the element x-axis and lies in the xv-plane with the positive direction lying in the same half plane as the positive v-axis. Finally, the element z-axis is obtained by using the right-hand rule. The xy-plane is known as *plane 1* and the xz-plane as *plane 2*.



6. Continue creating the bar elements by connecting the following nodes:

Nodes		Select
2	**3**	**OK**
3	**4**	**OK**

Property **prop_2**

Nodes		Select
1	**5**	**OK**
5	**6**	**OK**
6	**4**	**OK**
2	**5**	**OK**
3	**6**	**OK**

Select **Cancel**.

7. Create hinges at the ends of elements 4, 5, 6, 7, and 8:

Modify / Update Elements / Releases

Select elements **4**, **5**, **6**, **7**, **8** and, then, **OK**.

Define element Releases:

End A ☑ **RZ**

End B ☑ **RZ**

Select **OK**.

8. Create the model constraints.

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

Model / Constraint / Set

Title **constraint_1**

Select **OK**.

Now, define the relevant constraint for the model.

Model / Constraint / Nodal

Select **Node 1**

It will be marked with a white circle, a +1 will be added to the *Entity Selection* box, and you will be unable to highligh it anymore. These are all ways of checking which node you have selected.

Select **OK**.

On the *DOF* box, select

☑ **TX** ☑ **TY**

Select **OK.**

Notice that the constraint appears on the screen at Node 1, fixing the 1 and 2 directions (corresponding to TX and TY). Create the constraint for the other side of the model.

Select **Node 4 / OK**

On the *DOF* box, select

☑ **TY**

Select **OK**.

The rigid body motion must be removed. Do so by selecting **Nodes 1** and **4** / **OK**.

On the *DOF* box, select

☑ **TZ**

Select **OK**.

A warning message will appear: *Selected Constraints Already Exist. OK to Overwrite (No = Combine)?* Select **No** to combine.

Select **Node 1** / **OK**

On the *DOF* box, select

☑ **RX**

Select **OK**.

The same warning message above appears on the screen. Again, select **No** and, then, **Cancel**.

9. Create the model loading.

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

Model / **Load** / **Set** (or **Ctrl F2**)

Title	**load_1**

Select **OK**.

Define 2 tf point loads:

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

Select **Nodes 2** and **3 / OK**

Highlight **Force**

Method **Constant**

Load **FY** ✓ **-2**

Select **OK / Cancel**.

Define 1 tf/m distributed load:

Model / Load / Elemental

Select **Element 2 / OK**

Highlight **Distributed Load**

Method **Constant**

Load-End A **-1**

Load-End B **-1**

Select **OK**.

Direction ✓ **Global Y**

Select **OK / Cancel**.

10. Submit the model for analysis.

File / Analyze

Analysis Type **Static**

Loads ✓ **load_1**

Constraints ✓ **constraint_1**

✓ **Run Analysis**

Select **OK**.

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

Be sure to set the desirable working directory.

File Name	**work_2**

Select **Save**. When the MSC/NASTRAN manager is through running, MSC/NASTRAN for Windows will be restored on your screen, and 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. Then, select **Continue**.

11. List the results of the analysis.

List/ Output/ Query

Output Set	**1..MSC / NASTRAN Case 1**
Category	**Any Output**
Entity	**Node**
ID	**2**

Select **OK**.

Double click at the bottom of the screen to see the results. Double click again to return.

What is the displacement at grid (node) 2?

Disp. X = ___________________

Disp. Y = ___________________

Disp. Z = ___________________

What is the constraint force at grid (node) 4?

Force X = ___________________

Force Y = ___________________

Force Z = ___________________

What are the internal forces acting on element 5?

Axial force = ___________________

Shear force = ___________________

Bending moment = ___________________

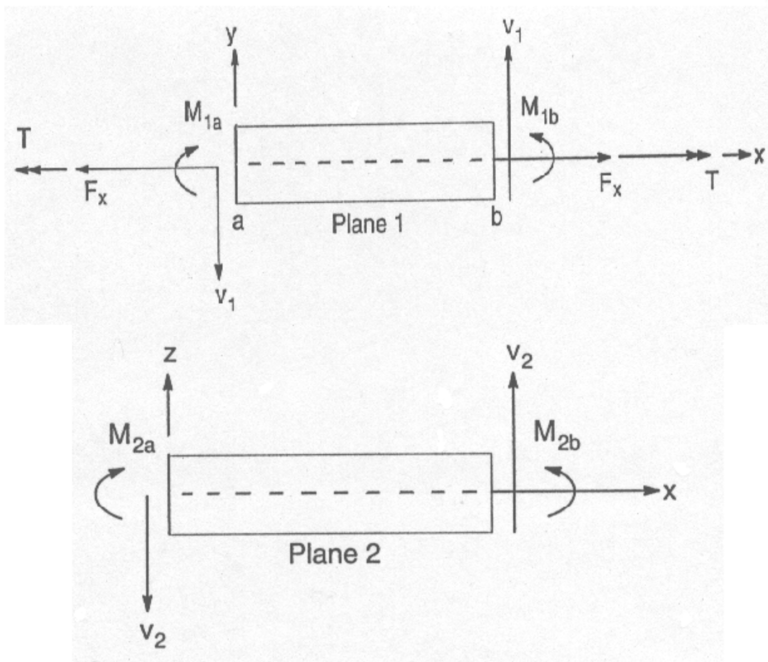
What are the internal forces acting on the left of grid (node) 3?

Axial force = ___________________

Shear force = ___________________

Bending moment = ___________________

NOTE: The element internal forces in planes 1 and 2 are given in the figure:



12. Display the deformed plot on the screen.

    Before displaying the deformed plot on the screen, you may want to remove the load and boundary constraint markers.

View / Options / Quick Options (or **Ctrl Q**)

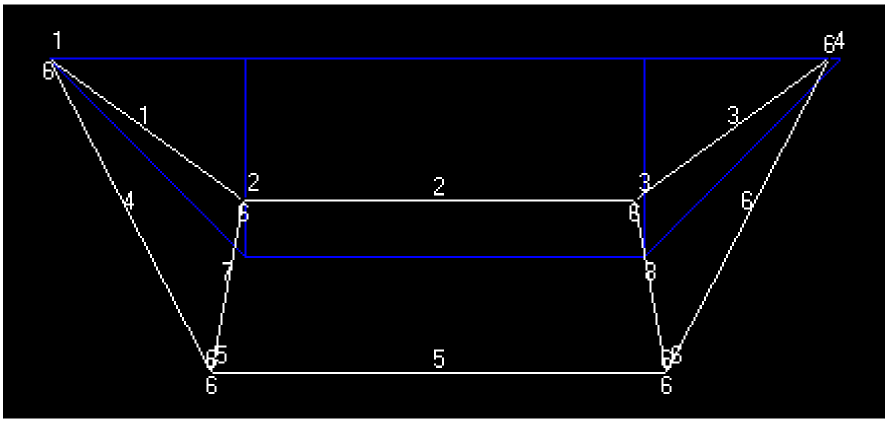
☐ **Force** / ☐ **Distributed Load** / ☐ **Constraint** / **Done** / **OK**

Plot the deformation of the frame.

View/ Select (or **F5**)

Deformed Style **Deform**

Select **Deformed and Contour Data / OK / OK**.



As the deformed plot is obtained by connecting the displaced nodes with straight lines, it may be improved by refining the mesh in the portion of the frame under bending.

13. Refine the mesh by subdividing the elements 1, 2 and 3 into four equal elements.

First, return the model to the original display:

View / Select (or **F5**)

Deformed Style **None - Model Only**

Select **OK**

Mesh / Remesh / Refine

Select **Elements 1**, **2** and **3** / **OK**.

Refinement ratio **4**

Select **OK**.

View / Options / Quick Options (or **Ctrl Q**)

✓ **Force** / ✓ **Distributed Load / Done / OK**

View / Regenerate (or **Ctrl G**).

Update the distributed load:

Model / Load / Elemental

Select **Elements 12**, **13** and **14** / **OK**.

Highlight **Distributed Load**

Method **Constant**

Load-End A **-1**

Load-End B **-1**

Select **OK**.

Direction ✓ **Global Y**

Select **OK** / **Cancel**.

14. Submit the model for analysis.

File / Analyze

Analysis Type **Static**

Loads ✓ **load_1**

Constraints ✓ **constraint_1**

✓ **Run Analysis**

Select **OK**.

When asked if you wish to save the model, respond **Yes** / **Continue**.

Does the refinement change the answer to the previous questions?

15. Display the deformed plot on the screen.

Before displaying the deformed plot on the screen, you may want to remove the load and boundary constraint markers.

View / Options / Quick Options (or **Ctrl Q**)

☐ **Force** / ☐ **Distributed Load / Done / OK**

Plot the deformation of the frame.

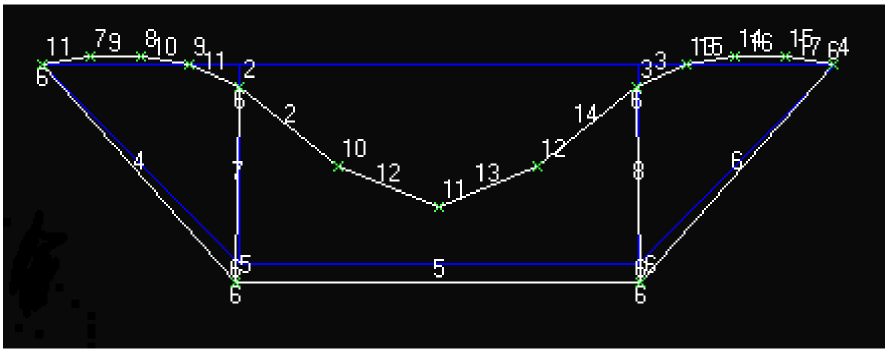
View/ Select (or **F5**)

Deformed Style	**Deform**

Select **Deformed and Contour Data**

Output Set	**2..MSC / NASTRAN Case 1**

Select **OK / OK**.



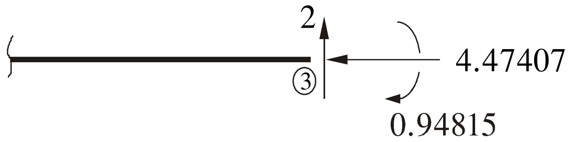
This concludes the exercise.

File / Save

File / Exit.

Answer

node 2	**disp. X**	-0.00055235
	disp. Y	-0.025108
	disp. Z	0
node 4	**force X**	0
	force Y	4
	force Z	0
element 5	**axial force**	4.47407
	shear force	0
	bending moment	0



NOTE: The above answer is not changed with the mesh refinement.