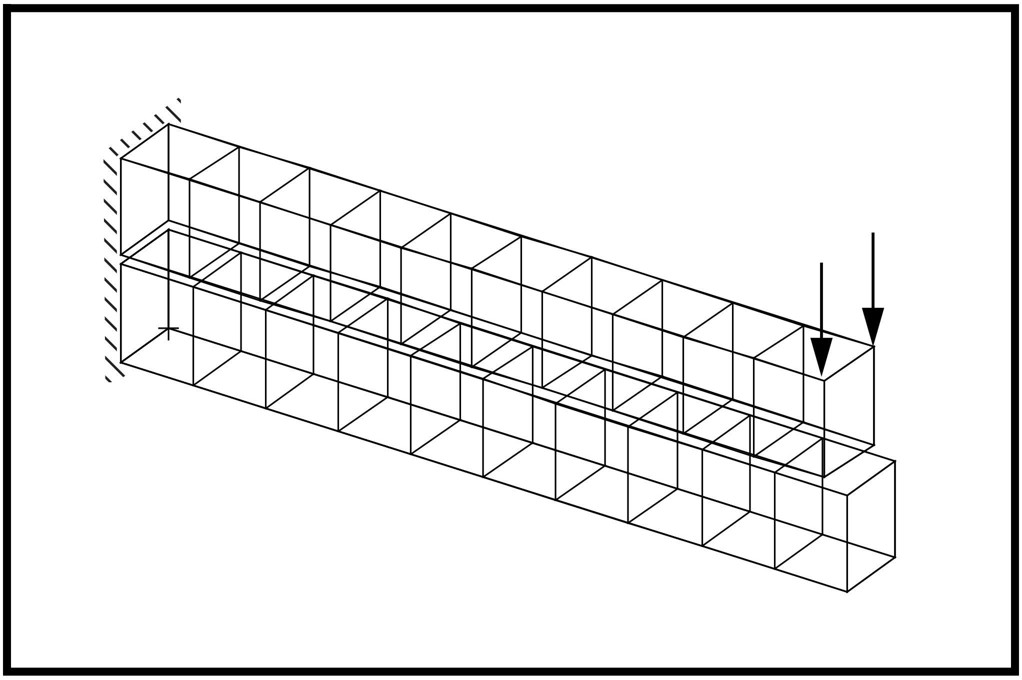
WORKSHOP PROBLEM 10a

Linear Static Analysis for a 3-D Slideline Contact



Objectives:

- Demonstrate the use of slideline contact.
- Run an MSC/NASTRAN linear static analysis.
- Create an accurate deformation plot of the model.
- Understand the linear behavior of the model.

Model Description:

Figure 10a.1 below shows a model of two 3-D cantilever beams. For this model the positive x-direction will be along the wall. There is a gap of 0.1 inches between the two beams where the top beam is shorter than the bottom. A total load of 2400 lb is placed at the edge of the shorter beam. The slideline master region will be the bottom of the top beam while the slave region will be the top of the bottom beam. Table 10a.1 below displays all the necessary dimensions and properties to complete the analysis.

Figure 10a.1 - 3-D Cantilever Beams

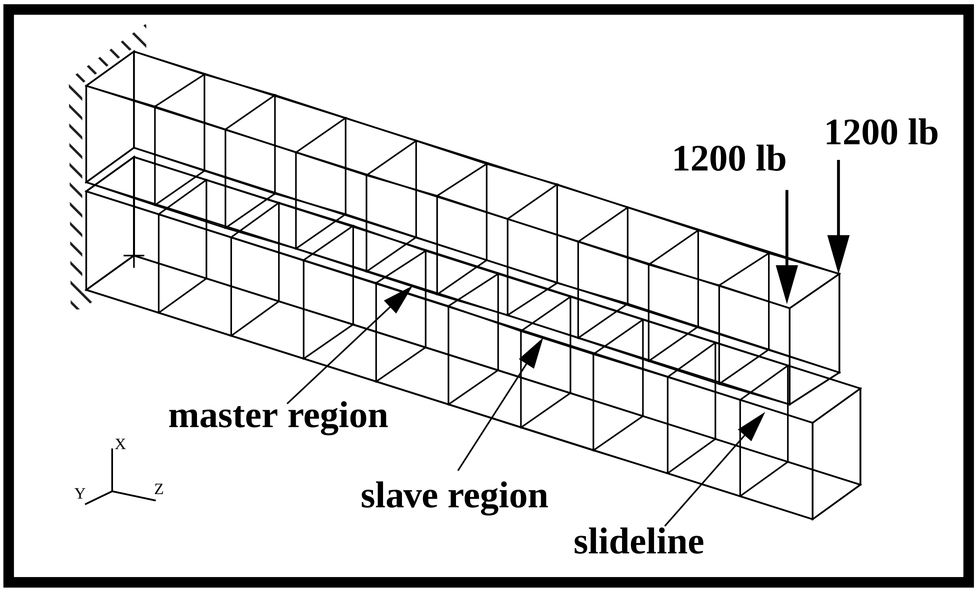


Table 10a.1 - Cantilever Beam Properties:

Dim. Top Beam:	**9.7L x 1H x 1W inches**
Dim. Bottom Beam:	**10 L x 1H x 1W inches**
Gap Between Beams:	**0.1 inches**
Elastic Modulus:	**10E+06**
Elastic Modulus:	**0.33**
Coefficient of Friction:	**0.1**
Total Force on Top Beam:	**2400 lb**

Exercise Procedure:

1. Start up MSC/NASTRAN for Windows V3.0 and begin to create a new model.

Double click on the icon labeled MSC/NASTRAN for Windows V3.0.

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

Open Model File: **New Model**

(Optional) For users who wish to remove the default rulers in the work plane model, please do the following:

View/Options...

● **Tools and View Style**

Category: **Workplane and Rulers**

Draw Entity ☐

Apply

Cancel

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

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

Model/Material...

Title: **mat_1**

Youngs Modulus, E: **10E+06**

Poisson's Ratio, nu: **0.33**

OK

Cancel

3. Create a property called **prop_1** for the solid elements of the model.

Model/Property...

Title: **prop_1**

Material: **1..mat_1**

Elem/Property Type...

Volume Elements: ● **Solid**

OK

OK

Cancel

4. Create the geometry for the 2 beams.

Mesh/Between...

To select the property, click on the list icon next to the databox and select **prop_1**.

Property: **1..prop_1**

Mesh size/ # Nodes/ Dir 1: **2**

Mesh size/ # Nodes/ Dir 2: **2**

Mesh size/ # Nodes/ Dir 3: **11**

OK

	X:	*Y:*	*Z:*
Corner 1:	**0**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**1**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 3:	**1**	**1**	**0**

OK

	X:	*Y:*	*Z:*
Corner 4:	**0**	**1**	**0**

OK

Continue the same procedure for the remaining four nodes for the bottom beam.

X	Y	Z
0	0	10
1	0	10
1	1	10
0	1	10

Now to create the second beam using the above procedure.

To select the property, click on the list icon next to the databox and select **prop_1**.

Property: **1..prop_1**

Mesh size/ # Nodes/ Dir 1: **2**

Mesh size/ # Nodes/ Dir 2: **2**

Mesh size/ # Nodes/ Dir 3: **11**

OK

	X:	*Y:*	*Z:*
Corner 1:	**1.1**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**2.1**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 3:	**2.1**	**1**	**0**

OK

	X:	*Y:*	*Z:*
Corner 4:	**1.1**	**1**	**0**

OK

Continue the same procedure for the remaining four nodes for the

top beam.

X	Y	Z
1.1	0	9.7
2.1	0	9.7
2.1	1	9.7
1.1	1	9.7

To fit the display onto the screen, use the Autoscale feature.

View/Autoscale

The model can be viewed at the same angle as shown in Figure 10a.1 by doing the following:

View/Rotate...

	X:	*Y:*	*Z:*
View 1:	**30**	**60**	**80**

A following section of this exercise requires using the mouse to select individual nodes. You may want to increase the model size on your screen by magnifying it. This allows for a more distinct view of the nodes.

Mag...

Magnification Factor: **1.6**

OK

Alternatively, you may use the zoom icons located at the top of your screen. This will enable you to position the model on your screen as it appears similar to the Figure 10a.1.

5. Create a coordinate system.

This will be our slideline plane vector coordinate system.

Model/Coord Sys...

Title: **coord_1**

Method: ● **ZX Axes**

OK

	X:	*Y:*	*Z:*
Origin:	**1**	**0**	**0**

OK

Vector along CSys Z-Axis:

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

OK

Vector in CSys ZX-Plane:

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

OK

Cancel

For this new coordinate system, the positive y-direction is in the up direction.

6. Create a property called **prop_2** for the slide line element of the model.

Model/Property...

Title: **prop_2**

Change the property type from plate elements (default) to slide line element.

Elem/Property Type...

Other Elements: ● **Slide Line**

OK

Property Values /
Stiffness Scale Factor: **1**

Static Friction Coefficient: **0.1**

Slide Line Plane
(Coord Sys XY): **3..coord_1**

Property Values: ● **Symmetrical Penetration**

OK

Cancel

7. Create the slideline element.

NOTE: To get a better understanding of slideline contact, review the supplement provided at the end of this exercise.

Model/Element...

Property: **2..prop_2**

Master Nodes...

Using the mouse select the nodes on the bottom edge corner along the length of the shorter beam. Start from the node at the free end of the beam.

OK

Slave Nodes...

Select the nodes on the top edge corner along length of the longer beam. Be sure to pick the Slave nodes in the opposite direction from the order of which the Master nodes were chosen. Start from the node at the constrained end.

OK

OK

IF told that nodes should be selected in reversed order, answer **Yes**.

Yes

Before going on to the next step, repeat the above procedure exactly for the other edge along the same surface of the two beams. Start with the Master nodes.

IF you are told once again that the nodes should be selected in reversed order, answer **Yes** again and then click **Cancel**.

Yes

Cancel

8. Create the model constraint sets.

It is necessary to create a constraint set, before applying the constraint on the model.

Create the constraint set.

Model/Constraint/Set...

ID:	**1**
Title:	**constraint**

OK

Now define the end constraints for the model (the left side).

Model/Constraint/Nodal...

Select all nodes at the flushed end of both beams, Nodes 1 to 4, and Nodes 45 to 48.

OK

Click on the **Fixed** button.

Fixed

On the *DOF* box, it should appear as follows.

☒ **TX** ☒ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Cancel

To clean up the display on the screen, use the Redraw feature.

View/Redraw

9. Create the loading of the model.

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

Model/Load/Set...

ID:	**1**
Title:	**load_1**

OK

Apply the nodal load.

Model/Load/Nodal...

Select top two corner nodes of the shorter beam, Node 88 and 86.

OK

Coord Sys:	**0..Basic Rectangular**

Highlight **Force.**

FX ☒	**-1200**

OK

Cancel

10. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Type: **1..Static**

OK

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

File name: **prob10a**

Write

☒ **Run Analysis**

Advanced...

Problem ID: **Slideline contact**

OK

Under *Output Requests*, deselect everything except the following:

☒ **Displacement**

Also, change output request to:

Output Request: **2..Print and PostProcess**

Under *Analysis Case Requests*, enter the following:.

☒ **Loads =** **1..load_1**

☒ **Constraint =** **1..constraint_1**

OK

OK

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

Yes

File name: **prob10a**

Save

When the MSC/NASTRAN manager is through running, MSC/NASTRAN 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.

Continue

When asked if it is "OK to Begin Reading File C:\TEMP\prob10a.xdb", respond **Yes**.

Yes

11. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set:	**1..MSC/NASTRAN Case 1**
Category:	**1..Displacement**
Entity:	● **Node**
ID:	**88**
OK	*(Select a load application point.)*

NOTE: You may want to expand the message box in order to view the results. To do this, double click on the message box. Adjust the size of the box to your preference by dragging the top border downward.

What are the x and y displacements of Node 88 at the end of the first subcase?

T1= ____________________

T2= ____________________

12. Display the deformed plot on the screen.

First, you may want to remove the labels and LBC markers in order to give a better view of the deformation.

View/Options...

Quick Options...

Labels Off

☐ **Coordinate System**

☐ **Load - Force**

☐ **Constraint**

Done

OK

Plot the deformation of the structure.

View/Select...

Deformed Style:	● **Deform**
Contour Style:	● **Contour**

Deformed and Contour Data...

Data Selection/Category:	**1..Displacement**
Output Set:	**1..MSC/NASTRAN Case 1**
Output Vectors/Deformation:	**1..T1 Translation**
Output Vectors/Contour:	**1..T1 Translation**

OK

OK

In order to see the deformation results accurately, you will need to turn off the display scaling of the actual deformation.

View/Options...

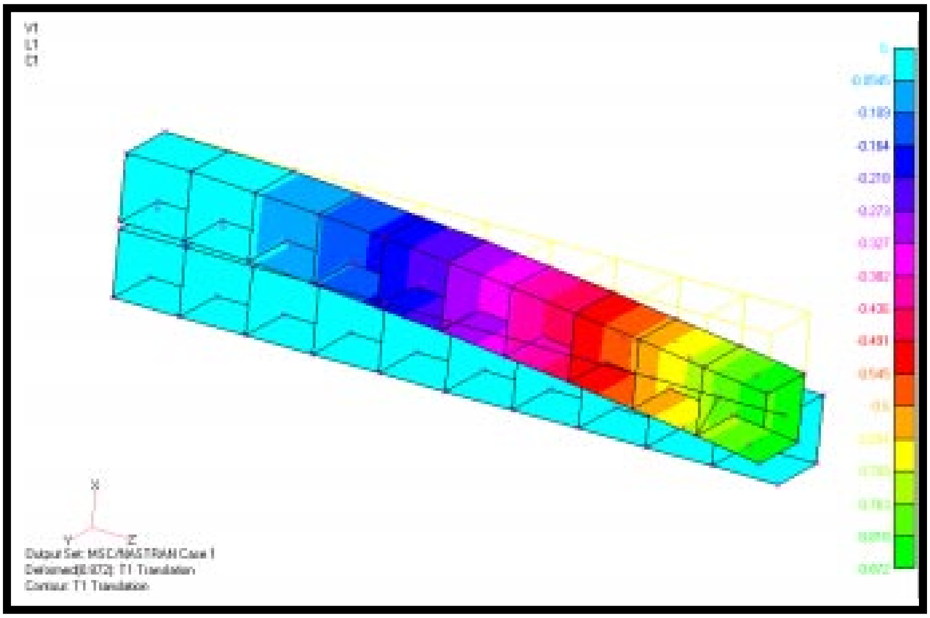
Category:	● **PostProcessing**
Options:	**Deformed Style**

% of Model (Actual) ☐

OK

The XY view should appear as follows:

Figure 10a.2:

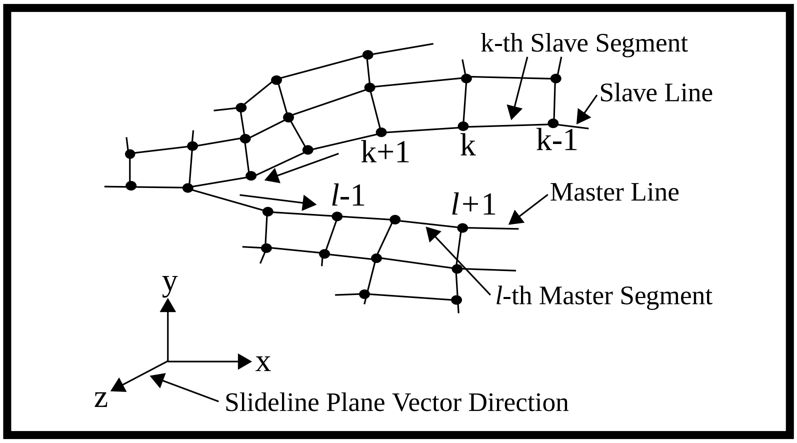


One can see the result obtained in this linear analysis is physically impossible (see Figure 10a.2). By intuition, the lower beam should deform with the upper beam once the two beams come into contact. The outcome of this linear analysis shows that a nonlinear analysis will be required in order to predict the correct deformation of this structure.

This concludes the exercise.

Slideline Contact Supplement:

Figure 10a.3: A Typical Finite Element Slideline Contact Region



- X-Y plane is the slideline plane. Unit normal in the Z-direction is the slideline plane vector.
- Arrows show positive direction for ordering nodes. Counter-clockwise from master line to slave line.
- Slave and master segment normals **MUST** face each other.
- As a rule of thumb, the master nodes should always penetrate the slave nodes. In another word, the master nodes should be associated with the moving object or the loaded object. The slave nodes should be associated with the fixed object. In this example, the master nodes were associated with the upper beam while the slave nodes were associated with the lower beam.

Adhering to these rules will keep your analysis free of FATAL ERRORS!

	T1	T2
Step 1	**-0.87236**	**0.0000041877**