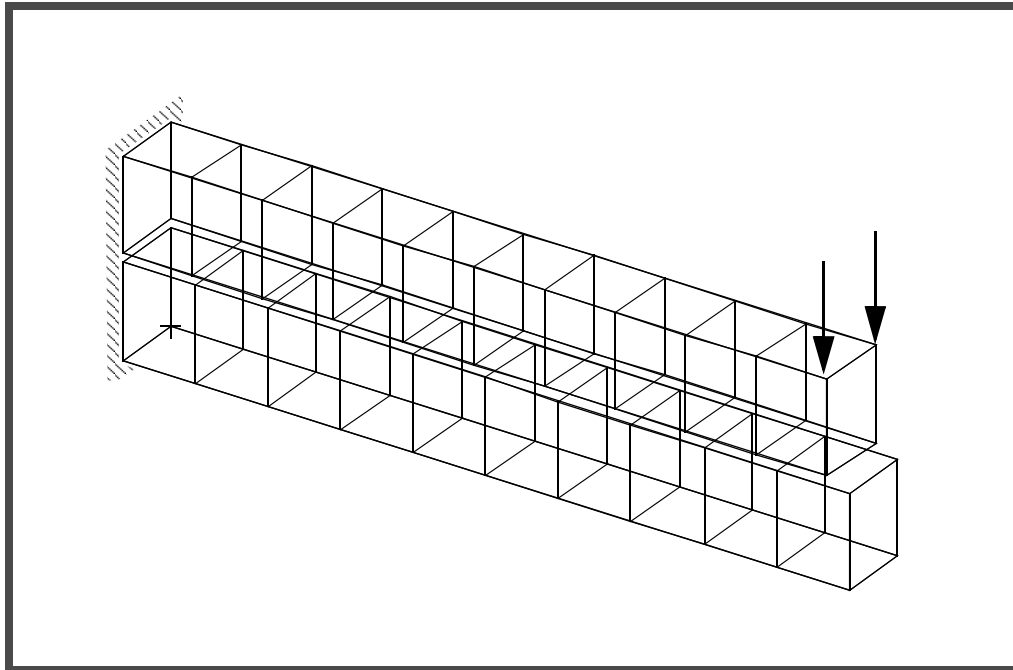

WORKSHOP 10a

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

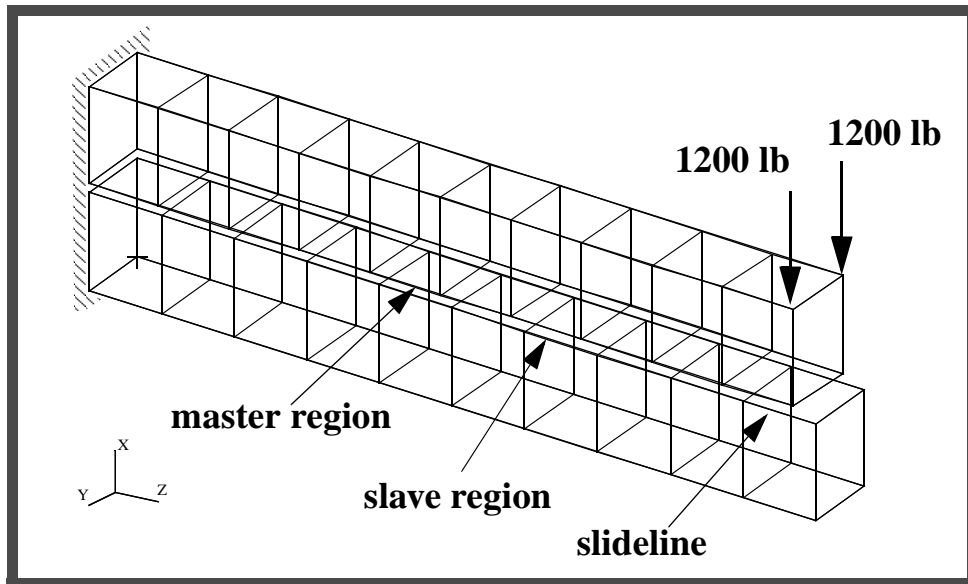


Table 10a.1 - Cantilever Beam Properties:

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 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

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

View/Options...

Category:

Tools and View Style

Options:

Workplane and Rulers

Draw Entity

OK

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.

<u>X</u>	<u>Y</u>	<u>Z</u>
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.

<u>X</u>	<u>Y</u>	<u>Z</u>
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:
<i>View 1:</i>	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

Origin:

X:	Y:	Z:
1	0	0

OK

Vector along CSys Z-Axis:

Base:

X:	Y:	Z:
1	0	0

Tip:

1	1	0
----------	----------	----------

OK

Vector in CSys ZX-Plane:

Base:

X:	Y:	Z:
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. Refer to Figure 10a.1 for illustration of master and slave regions.

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**.

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**.

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:

Title:

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.

Click on the **Fixed** button.

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...

<i>ID:</i>	<input type="text" value="1"/>
<i>Title:</i>	<input type="text" value="load_1"/>

OK

Apply the nodal load.

Model/Load/Nodal...

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

OK

<i>Coord Sys:</i>	<input type="text" value="0..Basic Rectangular"/>
-------------------	---

Highlight **Force**.

<i>FX</i> <input checked="" type="checkbox"/>	<input type="text" value="-1200"/>
---	------------------------------------

OK
Cancel

10. Submit the job for analysis.

File/Export/Analysis Model...

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

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

Draw:

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:

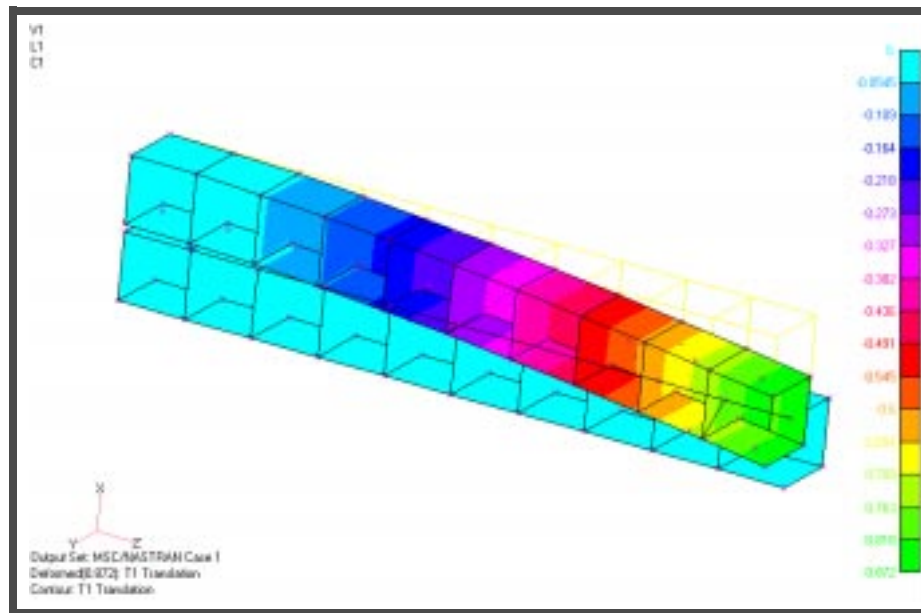
Deformed Style

% of Model (Actual)

OK

The XY view should appear as follows:

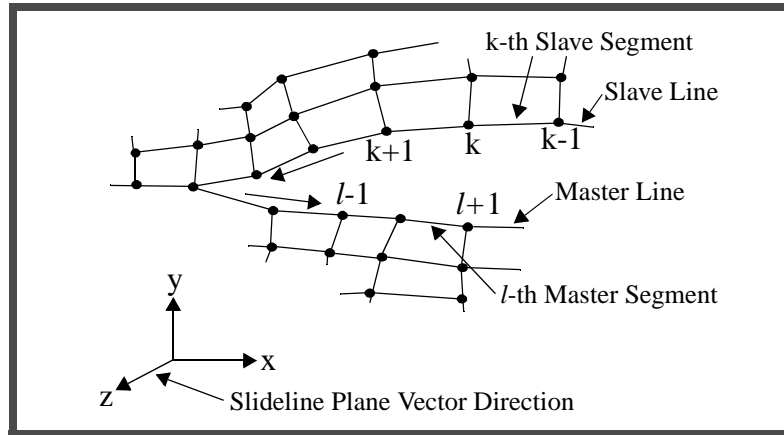
Figure 10a.2:



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!



Step 1	-0.87236	0.000041877
	T1	T2