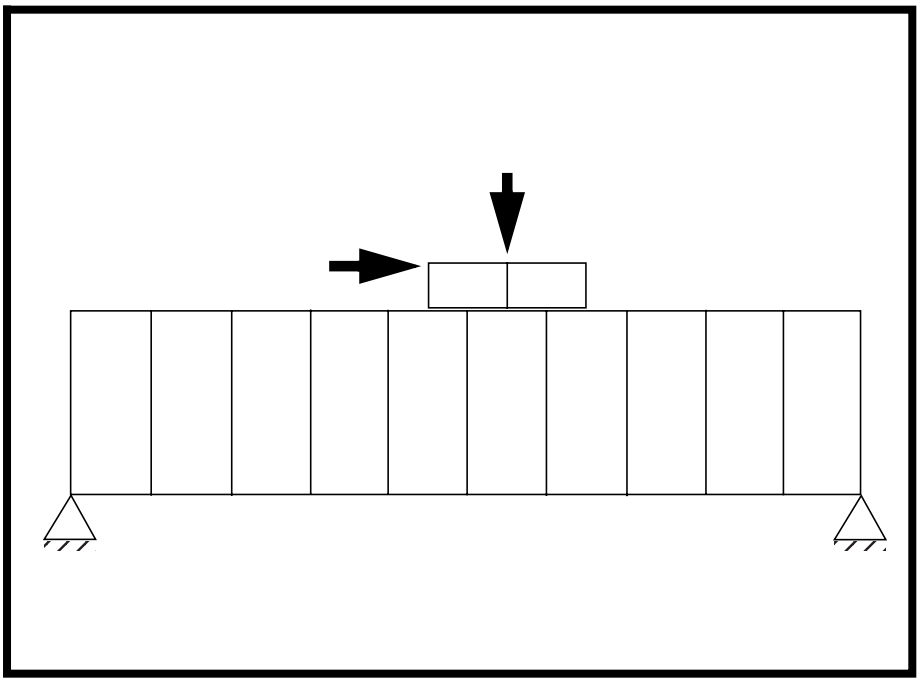
WORKSHOP PROBLEM 9

2-D Slideline Contact

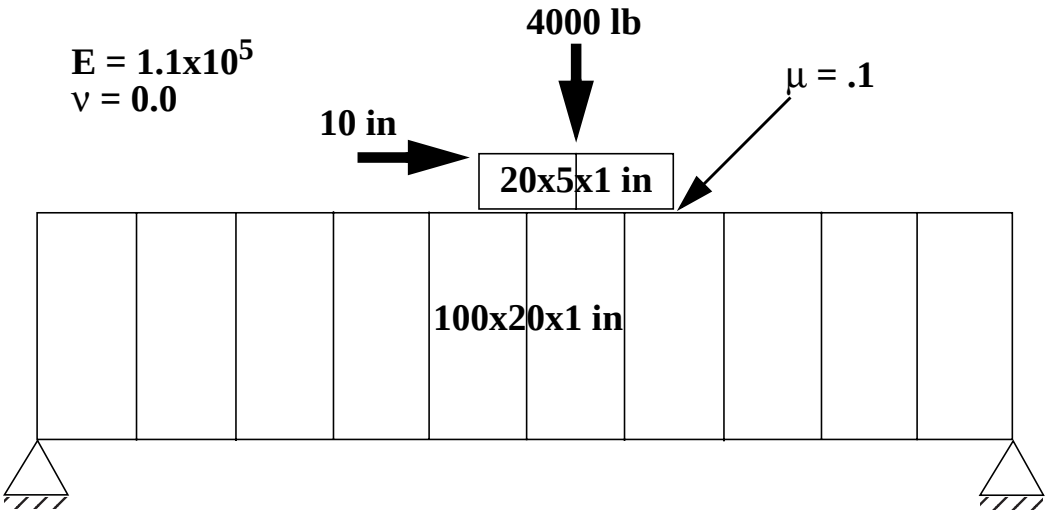


Objectives:

- Demonstrate the use of slideline contact.
- Create the appropriate load cases, one with enforced displacement and the other without.
- Run an MSC/NASTRAN nonlinear static analysis.
- Create an accurate deformation plot of all the subcases.

Model Description:

Figure 9.1 - The Structure, Material Properties, and Loading



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: **1e5**

OK

Cancel

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

Model/Property...

Title: **prop_1**

Material: **1..mat_1**

Elem/Property Type...

Plate Elements: ● **Membrane**

OK

Thicknesses, Tavg or T1: **1**

OK

Cancel

4. Create the NASTRAN geometry for the 2 plates.

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: **11**

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

OK

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

OK

	X:	*Y:*	*Z:*
Corner 2:	**100**	**0**	**0**

OK

	X:	*Y:*	*Z:*
Corner 3:	**100**	**20**	**0**

OK

	X:	*Y:*	*Z:*
Corner 4:	**0**	**20**	**0**

OK

Mesh/Between...

Mesh size/ # Nodes/ Dir 1: **3**

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

OK

	X:	*Y:*	*Z:*
Corner 1:	**45**	**20**	**0**

OK

	X:	*Y:*	*Z:*
Corner 2:	**65**	**20**	**0**

OK

	X:	*Y:*	*Z:*
Corner 3:	**65**	**25**	**0**

OK

	X:	*Y:*	*Z:*
Corner 4:	**45**	**25**	**0**

OK

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

View/Autoscale

5. Create a coordinate system.

Model/Coord Sys...

Title: **coord_1**

Method: ● **ZX Axes**

OK

	X:	*Y:*	*Z:*
Origin:	**45**	**20**	**0**

OK

Vector along CSys Z-Axis:

	X:	*Y:*	*Z:*
Base:	**45**	**20**	**0**
Tip:	**45**	**20**	**-1**

OK

Vector in CSys ZX-Plane:

	X:	*Y:*	*Z:*
Base:	**45**	**20**	**0**
Tip:	**46**	**20**	**0**

OK

Cancel

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: **10**

Static Friction Coefficient: **0.1**

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

Property Values: ● **Symmetrical Penetration**

OK

Cancel

7. Create the slideline element.

Model/Element...

Property: **2..prop_2**

Master Nodes...

Select the nodes on the bottom edge of the top surface, **Node 23 to 25**.

OK

Slave Nodes...

Select the nodes on the top edge of the bottom surface, **Node 12 to 22**.

OK

OK

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

Yes

Cancel

8. Create the model constraint sets.

Since the model will require two load cases with different constraints, it is necessary to create two sets of constraints.

First, create the first constraint set.

Model/Constraint/Set...

ID: **1**

Title: **constraint_1**

OK

Now define the end constraints for the model.

Model/Constraint/Nodal...

Select **all nodes**.

Select All
OK

On the *DOF* box, select these four boxes.

☐ **TX** ☐ **TY** ☒ **TZ**

☒ **RX** ☒ **RY** ☒ **RZ**

OK

Next, select all nodes on the bottom edge of the bottom surface, **Node 1 to 11**.

OK

On the *DOF* box, select the following boxes.

☒ **TX** ☒ **TY** ☐ **TZ**

OK

When asked ‘OK to Overwrite (NO = combine)’, select **NO**.

No
Cancel

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

View/Redraw

9. Create the second load set by combining the first constraint set and add in additional information.

Model/Constraint/Combine...

From Set: **1..constraint**

Last One

After combining the constraints from step 8, add in the following new constraints.

Model/Constraint/Nodal...

Select **Node 26, 27, and 28**.

OK

On the *DOF* box, select the TX box.

☒ **TX** ☐ **TY** ☐ **TZ**

OK

NOTE: This constraint in the x-direction is necessary to keep the model static when a nodal displacement loading in the x-direction is involved.

When asked “OK to Overwrite (NO = combine)”, select **No**.

No

Cancel

Refresh the display by using the Redraw option.

View/Redraw

10. 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

Define the options for load set 1 for a nonlinear analysis.

Model/Load/Nonlinear Analysis...

Solution Type: ● **Static**

Default...

Basic / Number of Increments: **1**

Stiffness Updates / Method: **1..AUTO**

Output Control / Intermediate: **1..YES**

☒ Load **1e-6**

☒ Work **1e-10**

OK

Apply the nodal load.

Model/Load/Nodal...

Select top corner nodes of top surface, **Node 26 and 28.**

OK

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

Highlight **Force.**

FY ☒ **-1000**

OK

Select top mid-node on top edge of top surface, **Node 27.**

OK

FY ☒ **-2000**

OK

Cancel

11. You will need to repeat **Step 10** to create the next nonlinear static load set. Use the following table to make the appropriate changes to the steps:

NOTE: Also be certain to change the ID or the first load set will be written over!

Model / Load / Set...	*ID*	**2**
	Title	**load_2**
Model / Load Nonlinear Analysis...	*Basic / Number of Increments:*	**10**

After inputting the vertical forces again on Node 26 to 28, do the following to add the enforced displacement.

Model/Load/Nodal...

Select **Node 26, 27, and 28**.

OK

Highlight **Displacement**.

TX ☒ **10**

OK

Cancel

12. Submit the job for analysis.

File/Export/Analysis Model...

Analysis Type: **10..Nonlinear Static**

OK

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

File name: **prob9**

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:

SUBCASE ID: **1**

☒ **Loads =** **1..load_1**

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

Write Case...

When you get confirmation that the subcase was written, click **OK**.

OK

SUBCASE ID: **2**

☒ **Loads =** **2..load_2**

☒ **Constraint =** **2..Combined Set**

OK

When you get confirmation that the subcase was written, click **OK**.

OK

OK

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

Yes

File name: **prob9**

Save

This analysis process will take longer than the other workshops. So do not stop the analysis if you see N4W repeat its analysis process.

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\prob9.xdb", respond **Yes**.

Yes

13. List the results of the analysis.

To list the results, select the following:

List/Output/Query...

Output Set: **22..Case 2 Step 1.000000**

Category: **1..Displacement**

Entity: ● **Node**

ID: **23**

OK

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 23 at the end of the first subcase?

T1= _____

T2= _____

What are the x and y displacements of Node 23 at the end

of the second subcase?

T1= _____

T2= _____

14. 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**

☐ **Load - Displacement**

☐ **Constraint**

Done

OK

Plot the deformation of the structure.

View/Select...

Deformed Style: ● **Deform**

Deformed and Contour Data...

Data Selection/Category: **1..Displacement**

Output Set: **40..Case 20 Step 2.000000**

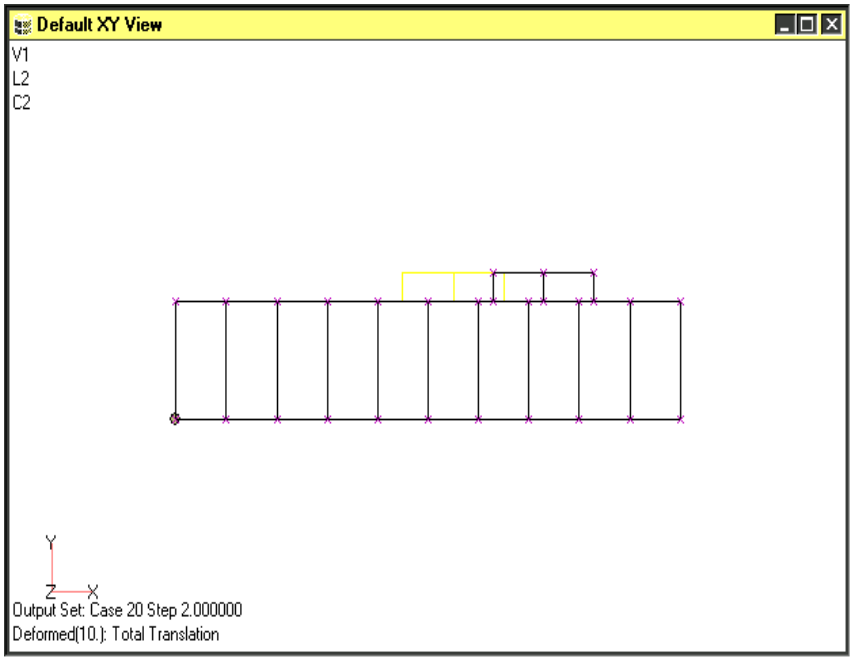
Output Vectors/Deformation: **1..Total Translation**

OK

OK

The XY view should appear as follows:

Figure 9.2:



This concludes the exercise.

	T1	T2
Step 1	0.00051589	-0.020228
Step 2	9.99372	-0.018511