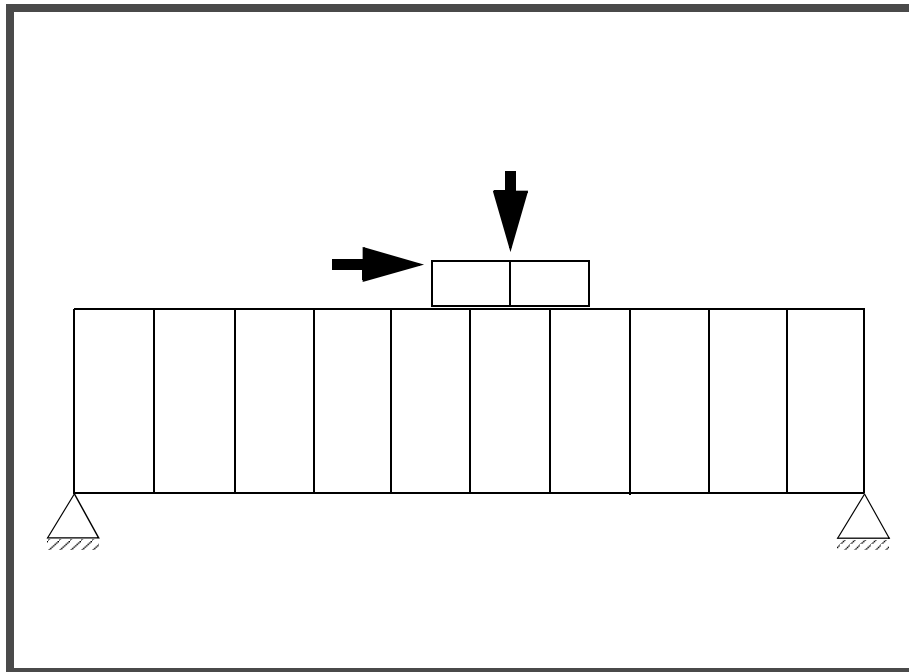

WORKSHOP PROBLEM 9

2-D Slideline Contact

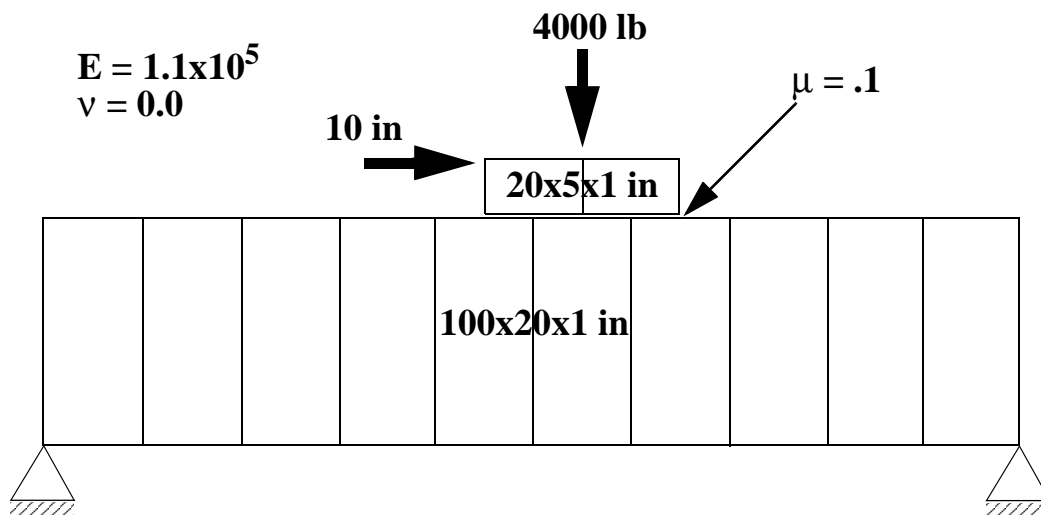


Objectives:

- Demonstrate the use of slideline contact.
- Create an accurate deformation plot of all the subcases to produce a false animation.

Model Description:

For the structure below:

**Add Case Control commands and Bulk Data Entries to:**

1. Model the contact between the block and the base.

Suggested Exercise Steps:

- Modify the existing MSC/NASTRAN input file by adding the appropriate nonlinear static analysis control parameters.
- Prepare the model for a nonlinear static analysis (SOL 106).
- Set up the appropriate subcase loading and analysis parameters (LOAD, NLPARAM)
- Input the necessary slideline contact parameters (BCONP, BFRIC, CORD2R, BLSEG).
- Generate an input file and submit it to the MSC/NASTRAN solver for a nonlinear static analysis.
- Review the results.

Input File for Modification:**prob9.dat**

```

ID NAS103 WORKSHOP 9
TIME 300
SOL 106
CEND
$
TITLE = SLINE2S: SYMMETRIC ELASTIC PUNCH WITH FRICTION
$
DISP = ALL
SUBCASE 1          $ VERTICAL LOAD
  LOAD = 1
  NLPARM = 410
SUBCASE 2    $ DISPLACEMENT TO THE RIGHT
  LOAD = 1
$
BEGIN BULK
PARAM,POST,-1
$
$ GEOMETRY
$
GRID,1,,0.,0.,0.,,123456 $
=,*1,,*(10.),==$
=9 $
GRID,12,,0.,20.,0.,,3456 $
=,*1,,*(10.),==$
=9 $
GRID,23,,45.,20.,0.,,3456 $
GRID,24,,55.,20.,0.,,3456 $
GRID,25,,65.,20.,0.,,3456 $
GRID,26,,45.,25.,0.,,3456 $
GRID,27,,55.,25.,0.,,3456 $
GRID,28,,65.,25.,0.,,3456 $
$
$ ELEMENTS
$
CQUAD4,1,1,1,2,13,12 $
=,*1,=*1,*1,*1,*1 $
=8 $
CQUAD4,11,1,23,24,27,26 $
=,*1,=*1,*1,*1,*1 $

```

```
PSHELL, 1, 1, 1., -1 $
MAT1, 1, 1.E5, , 0.0 $
$
$ PUNCH LOAD: VERTICAL LOAD
$
FORCE, 1, 26, , -1000., 0., 1., 0. $
FORCE, 1, 27, , -2000., 0., 1., 0. $
FORCE, 1, 28, , -1000., 0., 1., 0. $
$
$ LOAD FOR SUBCASE 2 : RIGHT HORIZONTAL DISPLACEMENT
$
$
$ SLIDELINE CONTACT
$
$
$ NONLINEAR SOLUTION STRATEGY: AUTO METHOD WITH DEFAULTS
$
NLPARM, 410, 1, , AUTO, , , PW, YES, +NLP41 $
+NLP41, , 1.E-6, 1.E-10 $
$
ENDDATA
```

Exercise Procedure:

1. Users who are not utilizing MSC/PATRAN for generating an input file should go to Step 15, otherwise, proceed to step 2.

2. Create a new database called **prob9.db**.

File/New...

New Database Name:

prob9

OK

In the **New Model Preference** form set the following:

Tolerance:

Default

Analysis Code:

MSC/NASTRAN

Analysis Type:

Structural

OK

3. Those who do not wish to set up the model themselves may want to play the session file, **prob9.ses**. If you choose to build the model yourself, proceed to the step 4.

File/Session/Play...

Session File List:

prob9.ses

Apply

The model has now been created. Skip to **Step 11**.

4. Create surfaces to represent the base and the block.

◆ Geometry

Action:

Create

Object:

Surface

Method:

XYZ

Vector Coordinate List:

<100, 20, 0>

Apply

Vector Coordinate List:

<20, 5, 0>

Origin Coordinates List:

[45, 20, 0]

Apply

5. Plant the mesh seed for mesh control.

◆ **Finite Elements**

Action:

Create

Object:

Mesh Seed

Type:

Uniform

Number =

1

Curve List:

(Select left edge of both surfaces.)

Apply

Number =

2

Curve List:

(Select top edge of top surface.)

Apply

Number =

10

Curve List:

(Select bottom edge of bottom surface.)

Apply

6. Mesh the model with Quad4 elements.

Action:

Create

Object:

Mesh

Type:

Surface

Element Topology:

Quad4

Surface List:

(Select both surfaces.)

Apply

7. Create the material property for the model.

◆ **Materials**

<i>Action:</i>	Create
<i>Object:</i>	Isotropic
<i>Method:</i>	Manual Input
<i>Material Name:</i>	mat_1
Input Properties...	
<i>Elastic Modulus =</i>	1e5
<i>Poisson's Ratio=</i>	0.0
Apply	
Cancel	

8. Create the element property for the model.

◆ **Properties**

<i>Action:</i>	Create
<i>Dimension:</i>	2D
<i>Type:</i>	Shell
<i>Property Set Name</i>	prop_1
Input Properties...	
<i>Material Name</i>	m:mat_1
<i>Thickness</i>	1
OK	
<i>Select Members</i>	<i>(Select both surfaces.)</i>
Add	
Apply	

9. Create the boundary conditions for the model.

Using the following icon increase the node size to make the node more visible for picking.



Node Size

Create the nodal degree of freedom constraint for the entire model.

◆ **Loads/BCs**

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name:

entire_model

Input Data...

Translations < T1 T2 T3 >

< , , 0 >

Rotations < R1 R2 R3 >

< 0, 0, 0 >

OK

Select Application Region...

Geometry Filter:

FEM

Select Nodes:

(Select all nodes in model.)

Add

OK

Apply

To clean up the display, use the following main menu icons:



Reset Graphics Refresh Graphics

Create the second model constraint.

New Set Name:

base_fixed

Input Data...

Translation < T1 T2 T3 >

< 0, 0, >

Rotations < R1 R2 R3 >:

< , , >

OK

Select Application Region...

Geometry Filter

Select Nodes

FEM

(Select nodes on bottom edge of bottom surface.)

Add

OK

Apply

10. Create the loading for the model.

Create the first load as follows:

◆ Loads/BCs

Action:

Create

Object:

Force

Type:

Nodal

New Set Name

load_1

Input Data...

Force < F1 F2 F3 >

<0, -1000, 0>

OK

Select Application Region...

Geometry Filter

Select Nodes

FEM

(Select top corner nodes of top surface.)

Add

OK

Apply

Now create the second load.

New Set Name:

load_2

Input Data...

Force <F1 F2 F3>

<0, -2000, 0>

OK

Select Application Region...

Select Nodes:

(Select mid-node on top edge of top surface.)

Add

OK

Apply

This is where the session file ends.

11. Create the enforced displacement of the block.

◆ **Loads/BCs**

Action:

Create

Object:

Displacement

Type:

Nodal

New Set Name:

push

Input Data...

Translations < T1 T2 T3 >:

< 10, , >

Rotations < R1 R2 R3 >:

< , , >

OK

Select Application Region...

Select Nodes:

(Select nodes on top edge of top surface.)

Add

OK

Apply

12. Create the slideline contact condition for the model.

◆ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Contact"/>
<i>Type:</i>	<input type="text" value="Element Uniform"/>
<i>Option:</i>	<input type="text" value="Slide Line"/>
<i>Analysis Type:</i>	<input type="text" value="Structural"/>
<i>New Set Name:</i>	<input type="text" value="contact"/>

<i>Penetration Type:</i>	<input type="text" value="Symmetric"/>
<i>Friction Coefficient:</i>	<input type="text" value="0.1"/>
<i>Penalty Stiffness Scaling Factor:</i>	<input type="text" value="10"/>
<i>A Vector Pointing from Master to Slave Surface:</i>	<input type="text" value="< 0, -1, 0 >"/>

<i>Active Region:</i>	<input type="text" value="Master"/>
<i>Select Curves:</i>	<input type="text" value="(Select bottom edge of top surface.)"/>

<i>Active Region:</i>	<input type="text" value="Slave"/>
<i>Select Curves:</i>	<input type="text" value="(Select top edge of bottom surface.)"/>

13. Create the load cases for the model.

◆ **Load Cases**

<i>Action:</i>	<input type="text" value="Create"/>
----------------	-------------------------------------

Load Case Name:

case_1

Assign/Prioritize Loads/BCs

Select Loads/BCs to Add to Spreadsheet

Conta_contact
Displ_base_fixed
Displ_entire_model
Force_load_1
Force_load_2

OK

Apply

Load Case Name:

case_2

Assign/Prioritize Loads/BCs

Select Loads/BCs to Add to Spreadsheet:

Displ_push

OK

Apply

Take care to be sure that the LBC Scale Factor for each Load/BC in spreadsheet equal 1.

14. Generate an input file for the analysis.

Click on the **Analysis** radio button on the Top Menu Bar and set up the analysis as follows:

◆ **Analysis**

Action:

Analyze

Object:

Entire Model

Method:

Analysis Deck

Job Name

prob9

Solution Type...

Solution Type

● **NONLINEAR STATIC**

OK

Subcase Create...

Available Subcases:

case_1

Subcase Parameters...

Number of Load Increments:

Load Tolerance:

Work Tolerance:

OK

Output Requests...

Form Type:

Output Requests:

Delete

Intermediate Output Option:

OK

Apply

Repeat the above procedure for the second subcase.

Available Subcases:

Subcase Parameters...

Number of Load Increments:

Load Tolerance:

Work Tolerance:

OK

Output Requests...

Form Type:

Output Requests:

Delete

Intermediate Output Option:

OK

Apply

Cancel

Select both the subcases.

Subcase Select...

Subcases for Solution Sequence:

case_1
case_2

Subcases Selected:

(Deselect Default)

OK

Apply

An input file called **prob9.bdf** will be generated. This process of translating your model into an input file is called the Forward Translation. The Forward Translation is complete when the Heartbeat turns green. MSC/PATRAN users should now proceed to **Step 16**.

Generating an input file for MSC/NASTRAN Users:

15. MSC/NASTRAN users can generate an input file using the data from the Model Description. The result should be similar to the output below (**prob9.dat**):

```
ASSIGN OUTPUT2 = 'prob9.op2' , UNIT = 12
ID NAS103 WORKSHOP 9 SOLUTION
TIME 300
SOL 106
CEND
$
TITLE = SLINE2S: SYMMETRIC ELASTIC PUNCH WITH FRICTION
$
DISP = ALL
SUBCASE 1          $ VERTICAL LOAD
  LOAD = 1
  NLPARM = 410
SUBCASE 2    $ DISPLACEMENT TO THE RIGHT
  LOAD = 1
  NLPARM=420
  SPC=2
$
BEGIN BULK
PARAM,POST,-1
$
$ GEOMETRY
$
GRID,1,,0.,0.,0.,,123456 $
=,*1,,*(10.),== $
=9 $
GRID,12,,0.,20.,0.,,3456 $
=,*1,,*(10.),== $
=9 $
GRID,23,,45.,20.,0.,,3456 $
GRID,24,,55.,20.,0.,,3456 $
GRID,25,,65.,20.,0.,,3456 $
GRID,26,,45.,25.,0.,,3456 $
GRID,27,,55.,25.,0.,,3456 $
GRID,28,,65.,25.,0.,,3456 $
$
```

```

$ ELEMENTS
$
CQUAD4,1,1,1,2,13,12 $
=,*1,=*1,*1,*1,*1 $
=8 $
CQUAD4,11,1,23,24,27,26 $
=,*1,=*1,*1,*1,*1 $
PSHELL, 1, 1, 1., -1 $
MAT1, 1, 1.E5, , 0.0 $
$
$ PUNCH LOAD: VERTICAL LOAD
$
FORCE, 1, 26, , -1000., 0., 1., 0. $
FORCE, 1, 27, , -2000., 0., 1., 0. $
FORCE, 1, 28, , -1000., 0., 1., 0. $
$
$ LOAD FOR SUBCASE 2 : RIGHT HORIZONTAL DISPLACEMENT
$
SPC, 2, 26, 1, 10.0
SPC, 2, 27, 1, 10.0, 28, 1, 10.0
$
$ SLIDELINE CONTACT
$
BCONP, 10, 40, 50, , 10., 60, 2, 70
BFRIC, 60, , , 0.1
BLSEG, 40, 12, 13, 14, 15, 16, 17, 18, +BLSG1
+BLSG1, 19, 20, 21, 22
BLSEG, 50, 25, 24, 23
CORD2R, 70, , 0., 0., 0., 0., 0., 1., +CRD10
+CRD10, 1., 0., 0.
$
$ NONLINEAR SOLUTION STRATEGY: AUTO METHOD WITH DEFAULTS
$
NLPARM, 410, 1, , AUTO, , , PW, YES, +NLP41 $
+NLP41, , 1.E-6, 1.E-10 $
NLPARM, 420, 10, , AUTO, , , PW, YES, +NLP42
+NLP42, , 1.E-6, 1.E-10
$
ENDDATA

```

Submit the input file for analysis:

16. Submit the input file to MSC/NASTRAN for an analysis.
 - 16a. To submit the MSC/PATRAN **.bdf** file, find an available UNIX shell window. At the command prompt enter **nastran prob9.bdf scr=yes**. Monitor the analysis using the UNIX **ps** command.
 - 16b. To submit the MSC/NASTRAN **.dat** file, find an available UNIX shell window and at the command prompt enter **nastran prob9.dat scr=yes**. Monitor the analysis using the UNIX **ps** command.
17. When the analysis is completed, edit the **prob9.f06** file and search for the word **FATAL**. If no matches exist, search for the word **WARNING**. Determine whether the existing **WARNING** messages indicate any modeling errors.
 - 17a. While still editing **prob9.f06**, search for the word:

D I S P L A C E (spaces are necessary).

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= _____

Comparison of Results:

18. Compare the results obtained in the **.f06** file with the results on the following page:

LOAD STEP = 1.00000E+00

		D I S P L A C E M E N T V E C T O R						
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
.								
.								
.								
22	G	-3.213901E-03	2.042947E-03	0.0	0.0	0.0	0.0	
23	G	4.799817E-04	-1.948305E-02	0.0	0.0	0.0	0.0	
24	G	-9.838412E-05	-2.636322E-02	0.0	0.0	0.0	0.0	
25	G	-6.764003E-04	-1.952910E-02	0.0	0.0	0.0	0.0	
26	G	3.663775E-03	-3.048664E-02	0.0	0.0	0.0	0.0	
27	G	-8.672358E-05	-3.536061E-02	0.0	0.0	0.0	0.0	
28	G	-3.837358E-03	-3.053272E-02	0.0	0.0	0.0	0.0	

LOAD STEP = 2.00000E+00

		D I S P L A C E M E N T V E C T O R						
POINT ID.	TYPE	T1	T2	T3	R1	R2	R3	
.								
.								
.								
22	G	-4.396476E-03	1.590136E-03	0.0	0.0	0.0	0.0	
23	G	9.993721E+00	-1.851050E-02	0.0	0.0	0.0	0.0	
24	G	9.997439E+00	-2.663225E-02	0.0	0.0	0.0	0.0	
25	G	1.000035E+01	-2.080630E-02	0.0	0.0	0.0	0.0	
26	G	1.000000E+01	-2.875795E-02	0.0	0.0	0.0	0.0	
27	G	1.000000E+01	-3.563613E-02	0.0	0.0	0.0	0.0	
28	G	1.000000E+01	-3.255428E-02	0.0	0.0	0.0	0.0	

19. This ends the exercise for MSC/NASTRAN users. MSC/PATRAN Users should proceed to the next step.

20. Proceed with the Reverse Translation process, that is, importing the **prob9.op2** results file into MSC/PATRAN. To do this, return to the **Analysis** form and proceed as follows:

◆ **Analysis**

<i>Action:</i>	Read Output2
<i>Object:</i>	Result Entities
<i>Method:</i>	Translate
Select Results File...	
<i>Selected Results File:</i>	prob9.op2
OK	
Apply	

21. Post process the results from the analysis.

Also, erase all the geometry from the screen.

Display / Plot/Erase...

Erase All Geometry
OK

Clean up the viewport by using the following main menu icon:



Refresh Graphics

22. Next, bring up the **Results** form to create a false animation of the deformation by plotting the deformation for each load case.

Now we will generate the fringe plot of the model.

◆ **Results**

<i>Action:</i>	Create
<i>Object:</i>	Fringe

Now click on the **Select Results** icon.



Select Results

Select Result Case(s)

(Select all cases.)

Select Fringe Result

Displacements, Translational

Quantity:

Magnitude

Next click on the **Target Entities** icon.



Target Entities

Target Entity:

Current Viewport

Note: This feature allows you to view fringe plots of specific elements of your choice.

Click on the **Display Attributes** icon.



Display Attributes

Style:

Discrete/Smooth

Display:

Free Edges

Note: The **Display Attributes** form allows you the ability to change the displayed graphics of fringe plots.

Now click on the **Plot Options** icon.



Plot Options

Coordinate Transformation:

None

Scale Factor

1.0

Apply

The resulting fringe plot should display the displacement spectrum superimposed over the undeformed bar. The final fringe plot displaying the physical deformation of the model can be created as follows:

◆ **Results**

Action:

Object:

Now click on the **Select Results** icon.



Select Results

Select Result Case(s)

Select Deformation Result

Show As:

Click on the **Display Attributes** icon.



Display Attributes

In order to see the deformation results accurately, set the Scale Interpretation to True Scale with a Scale Factor of 1.

● *True Scale*

Scale Factor

■ *Show Undeformed*

Now click on the **Plot Options** icon.



Plot Options

Coordinate Transformation:

Scale Factor

Quit MSC/PATRAN when you have completed this exercise.

MSC/PATRAN .bdf file: prob9.bdf

```
$ NASTRAN input file created by the MSC MSC/NASTRAN input file
$ translator ( MSC/PATRAN Version 7.5 ) on January 15, 1998 at
$ 18:49:01.
ASSIGN OUTPUT2 = 'prob9.op2', UNIT = 12
$ Direct Text Input for File Management Section
$ Nonlinear Static Analysis, Database
SOL 106
TIME 600
$ Direct Text Input for Executive Control
CEND
SEALL = ALL
SUPER = ALL
TITLE = MSC/NASTRAN job created on 15-Jan-98 at 18:46:42
ECHO = NONE
MAXLINES = 999999999
$ Direct Text Input for Global Case Control Data
SUBCASE 1
$ Subcase name : case_1
  SUBTITLE=case_1
  NLPARAM = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : case_2
  SUBTITLE=case_2
  NLPARAM = 2
  SPC = 5
  LOAD = 5
  DISPLACEMENT(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
BEGIN BULK
PARAM POST -1
PARAM PATVER 3.
PARAM AUTOSPC YES
PARAM COUPMASS -1
PARAM K6ROT 100.
PARAM WTMASS 1.
PARAM LGDISP 1
PARAM,NOCOMPS,-1
```

```

PARAM PRTMAXIM YES
NLPARAM 1 1 AUTO 5 25 PW YES + A
* A 1.-6 1.-10 * B
* B
NLPARAM 2 10 AUTO 5 25 PW YES + C
* C 1.-6 1.-10 * D
* D

```

\$ Direct Text Input for Bulk Data

\$ Elements and Element Properties for region : prop_1

```

PSHELL 1 1 1.
CQUAD4 1 1 1 2 13 12
CQUAD4 2 1 2 3 14 13
CQUAD4 3 1 3 4 15 14
CQUAD4 4 1 4 5 16 15
CQUAD4 5 1 5 6 17 16
CQUAD4 6 1 6 7 18 17
CQUAD4 7 1 7 8 19 18
CQUAD4 8 1 8 9 20 19
CQUAD4 9 1 9 10 21 20
CQUAD4 10 1 10 11 22 21
CQUAD4 11 1 23 24 27 26
CQUAD4 12 1 24 25 28 27

```

\$ Referenced Material Records

\$ Material Record : mat_1

\$ Description of Material : Date: 21-Aug-97 Time: 11:49:06

```
MAT1 1 100000. 0.
```

\$ Nodes of the Entire Model

```

GRID 1 0. 0. 0.
GRID 2 10. 0. 0.
GRID 3 20. 0. 0.
GRID 4 30. 0. 0.
GRID 5 39.9999 0. 0.
GRID 6 49.9999 0. 0.
GRID 7 60. 0. 0.
GRID 8 70. 0. 0.
GRID 9 80. 0. 0.
GRID 10 90. 0. 0.
GRID 11 100. 0. 0.
GRID 12 0. 20. 0.
GRID 13 10. 20. 0.
GRID 14 20. 20. 0.
GRID 15 30. 20. 0.
GRID 16 39.9999 20. 0.
GRID 17 49.9999 20. 0.
GRID 18 60. 20. 0.

```

```

GRID 19      70. 20. 0.
GRID 20      80. 20. 0.
GRID 21      90. 20. 0.
GRID 22     100. 20. 0.
GRID 23      45. 20. 0.
GRID 24      55. 20. 0.
GRID 25      65. 20. 0.
GRID 26      45. 25. 0.
GRID 27      55. 25. 0.
GRID 28      65. 25. 0.
$ Loads for Load Case : case_1
SPCADD 2 4 6
LOAD 2 1. 1. 4 1. 6
$ Contact LBCs for Load Set : contact
BCONP 1 2 1 10. 1 2 1
BFRIC 1 .1
BLSEG 1 23 24 25
CORD2R 1 45. 20. 0. 45. 20. -1. + E
+ E 55. 20. 0.
BOUTPUT 1 23 24 25
BLSEG 2 22 21 20 19 18 17 16 + F
+ F 15 14 13 12
BOUTPUT 2 22 21 20 19 18 17 16 + G
+ G 15 14 13 12
$ Loads for Load Case : case_2
SPCADD 5 4 6 7
$ Enforced Displacements for Load Set : push
SPCD 5 26 1 10. 27 1 10.
SPCD 5 28 1 10.
LOAD 5 1. 1. 4 1. 6
$ Displacement Constraints of Load Set : entire_model
SPC1 6 3456 1 THRU 28
$ Displacement Constraints of Load Set : base_fixity
SPC1 4 12 1 THRU 11
$ Displacement Constraints of Load Set : push
SPC1 7 1 26 27 28
$ Nodal Forces of Load Set : load_1
FORCE 4 26 0 1000. 0. -1. 0.
FORCE 4 28 0 1000. 0. -1. 0.
$ Nodal Forces of Load Set : load_2
FORCE 6 27 0 2000. 0. -1. 0.
$ Referenced Coordinate Frames
ENDDATA c49a29de

```

