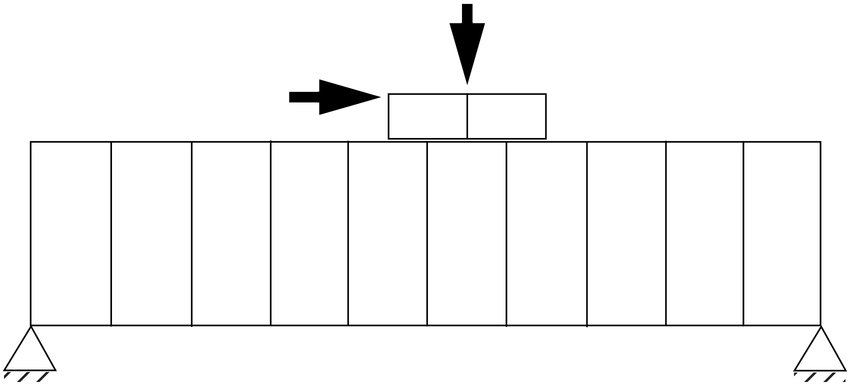
WORKSHOP PROBLEM 9

2-D Slideline Contact

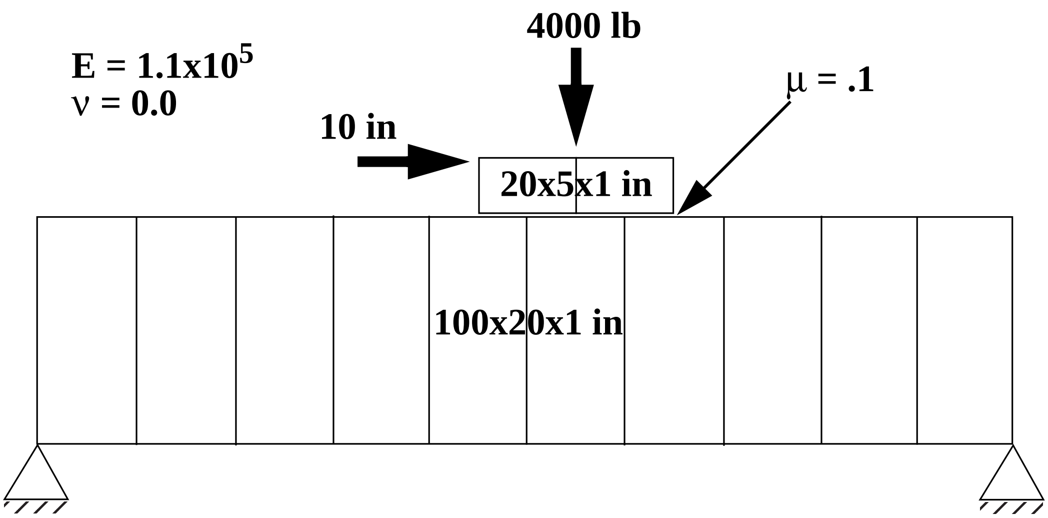


Objectives:

- 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, NLPARM)
- 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 utilitizing 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**

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

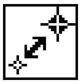
8. Create the element property for the model.

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**Shell**
Property Set Name	**prop_1**
Input Properties...	
Material Name	**m:mat_1**
Thickness	**1**
OK	
Select Members	(*Select both surfaces.)*
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.



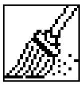
Node Size

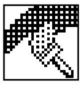
Create the nodal degree of freedom constraint for the entire model.

◆ **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	● **FEM**
Select Nodes	*(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	● **FEM**
Select Nodes	*(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**

Action:	**Create**
Object:	**Contact**
Type:	**Element Uniform**
Option:	**Slide Line**
Analysis Type:	**Structural**
New Set Name:	**contact**

Input Data...

Penetration Type:	**Symmetric**
Friction Coefficient:	**0.1**
Penalty Stiffness Scaling Factor:	**10**
A Vector Pointing from Master to Slave Surface:	**< 0, -1, 0 >**

OK

Select Application Region...

Active Region:	**Master**
Select Curves:	(*Select bottom edge of top surface.)*

Add

Active Region:	**Slave**
Select Curves:	(*Select top edge of bottom surface.)*

Add

OK

Apply

13. Create the load cases for the model.

◆ **Load Cases**

Action:	**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:	**1**
Load Tolerance:	**1e-6**
Work Tolerance:	**1e-10**
OK	
Output Requests...	
Form Type:	**Advanced**
Output Requests:	*(Select all but* ***DISPLACEMENT...****)*
Delete	
Intermediate Output Option:	**Yes**
OK	
Apply	

Repeat the above procedure for the second subcase.

Available Subcases:	**case_2**
Subcase Parameters...	
Number of Load Increments:	**10**
Load Tolerance:	**1e-6**
Work Tolerance:	**1e-10**
OK	
Output Requests...	
Form Type:	**Advanced**
Output Requests:	*(Select all but* ***DISPLACEMENT...****)*
Delete	
Intermediate Output Option:	**Yes**
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**

Action:	**Read Output2**
Object:	**Result Entities**
Method:	**Translate**

Select Results File...

Selected Results File:	**prob9.op2**

OK

Apply

21. Post process the results from the analysis.

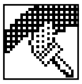
Also, erase all the geometry from the screen.

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**

Action:	**Create**
Object:	**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**

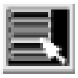
Apply

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:	**Create**
Object:	**Deformation**

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

 **Select Results**

Select Result Case(s)	*(Select all cases.)*
Select Deformation Result	**Displacements, Translational**
Show As:	**Resultant**

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	**1.0**

■ *Show Undeformed*

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

 **Plot Options**

Coordinate Transformation:	**None**
Scale Factor	**1.0**

Apply

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
  NLPARM = 1
  SPC = 2
  LOAD = 2
  DISPLACEMENT(SORT1,REAL)=ALL
$ Direct Text Input for this Subcase
SUBCASE 2
$ Subcase name : case_2
  SUBTITLE=case_2
  NLPARM = 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
NLPARM  1      1          AUTO    5     25    PW     YES   +    A
*     A           1.-6        1.-10                 *    B
*     B
NLPARM  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
```