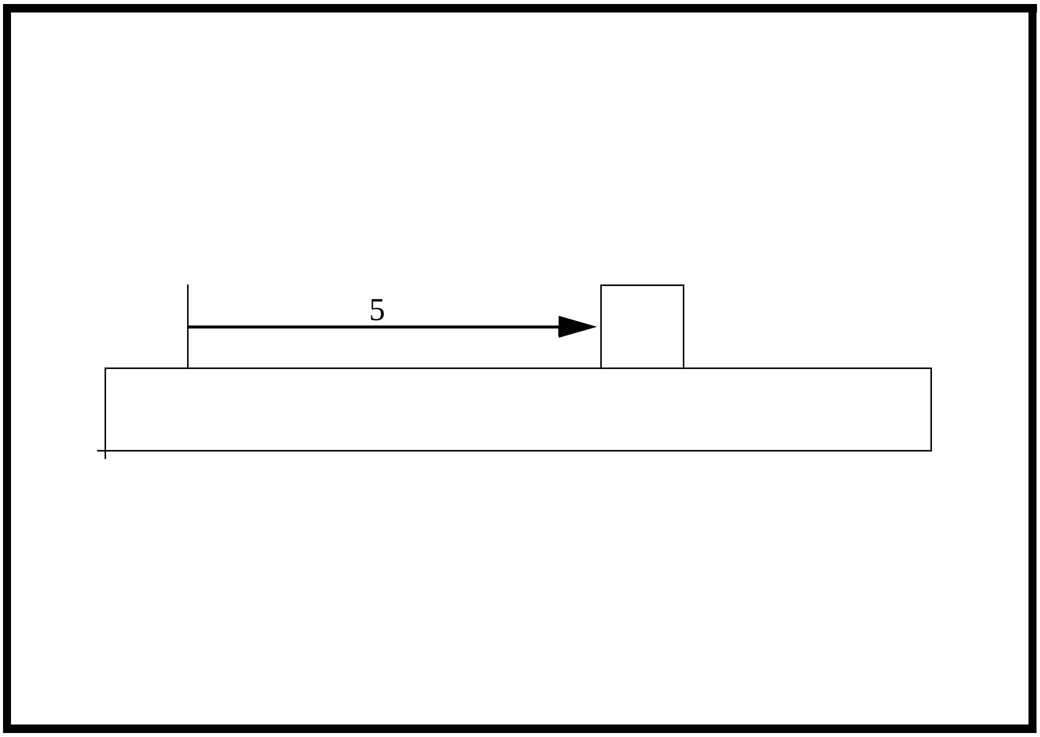
APPENDIX 2

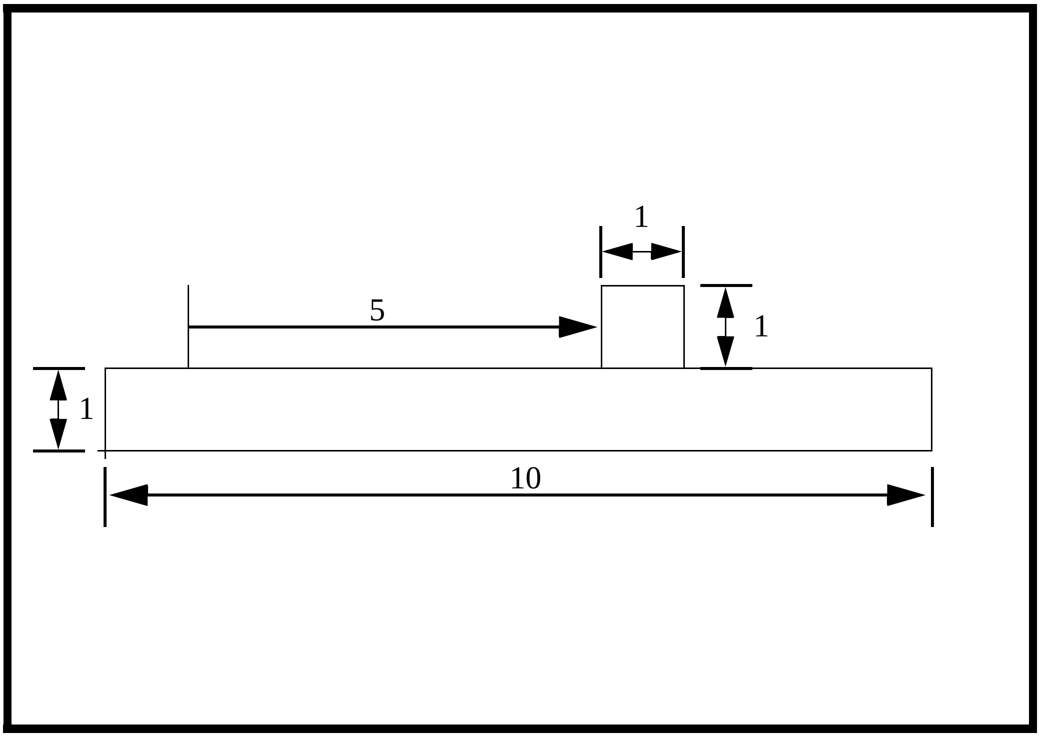
Sliding Block



Objectives:

- Demonstrate the use of Contact LBCs in a simple exercise.
- Present method for monitoring a non-linear analysis' progress.

Model Description:



As shown in the figure above, this exercise consists of a small block sitting on a baseplate. A pressure is applied to the top of the block, and it is moved 5 units to the right.

The purpose of this exercise is to introduce you to setting up a problem involving a Contact boundary condition. In addition, it gives a method for monitoring the progress of a nonlinear analysis.

Exercise Procedure:

1. Open a new database. Name it **slide_block.db**.

Type **p3** in your xterm. The *Main Window* and *Command Window* will appear.

File/New ...

New Database Name: **slide_block.db**

OK

The viewport (PATRAN's graphics window) will appear along with a *New Model Preference* form. The *New Model Preference* sets all the code specific forms and options inside MSC/PATRAN.

In the *New Model Preference* form set the *Analysis Code* to **MSC/ADVANCED_FEA.**

Tolerance:	◆ **Default**
Analysis Code:	**MSC/ADVANCED_FEA**
Analysis Type:	**Structural**

OK

2. Create the model geometry.

◆ **Geometry**

Action:	**Create**
Object:	**Surface**
Method:	**XYZ**
Vector Coordinate List:	**<10, 1, 0>**

Apply

Vector Coordinate List:	**<1, 1, 0>**
Origin Coordinate List:	**[1, 1, 0]**

Apply

Group/Create ...

New Group Name:	**geom**
Group Contents:	**Add All Geometry**

Apply

3. Create a mesh for the block and base.

New Group Name:	**fem_block**
Group Contents:	**Add Entity Selection**

Apply

Cancel

◆ **Finite Elements**

Action:	**Create**
Object:	**Mesh Seed**
Method:	**One Way Biased**
Number of Elements:	**4**
L2 / L1 =	**-2**
Curve List:	select the bottom

Action:	**Create**
Object:	**Mesh**
Method:	**Surface**
Global Edge Length:	**.25**
Mesher:	◆ **IsoMesh**
Surface List:	select the block (top)

Apply

Group/Create ...

New Group Name:	**fem_base**

Apply

Cancel

Global Edge Length:	**.15**
Mesher:	◆ **IsoMesh**
Surface List:	select the base (bottom)

Apply

What you have just done is meshed both the block and the base, and assigned each mesh to its own group. Now, create a group for all the FEM.

Group/Create ...

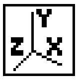
New Group Name:	**fem_all**
Group Contents:	**Add All FEM**

Apply

Cancel

4. Make sure that the element normals point in the positive z-direction.

To get a better view of the normals' direction, use the following toolbar icon:



Iso 1 View

Action:	**Verify**
Object:	**Element**
Test:	**Normals**
Display Control:	◆ **Draw Normal Vectors**

Apply

All of the vectors should be pointing in the positive z-direction already. If for some reason they are not, you can correct this by doing the following:

NOTE: You only need to do this if the normals are pointing in the wrong direction!

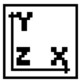
Action:	**Modify**
Object:	**Element**
Method:	**Reverse**
Element List:	select all incorrectly oriented elements

Apply

If you didn’t need to reverse any elements, clear up the screen by pressing:

Reset Graphics

Change the view back to the default by using the following toolbar icon:



Front View

5. Create two materials, **steel** and **aluminum**, to be used in the model.

Create the first material.

◆ **Materials**

Action:	**Create**
Object:	**Isotropic**
Method:	**Manual Input**
Material Name:	steel

Input Properties...

Elastic Modulus:	30E6
Poisson’s Ratio:	.3

Apply

Cancel

Create the second material.

Material Name:	aluminum

Input Properties...

Elastic Modulus:	10E6
Poisson’s Ratio:	.33

Apply

Cancel

6. Create and assign properties for both the block and the base.

First, lets deal with the block.

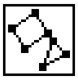
Group/Post ...

Select Groups to Post:	**fem_block**
Apply	
Cancel	

◆ **Properties**

Action:	**Create**
Dimension:	**2D**
Type:	**2D Solid**
Property Set Name:	**block**
Input Properties...	
Material Name:	**steel**
Thickness:	**1.0**
OK	

In order to select the elements on the screen, be sure to use the following entity select icon:

 **2D Element**

Select Members:	select all on screen
Add	
Apply	

Post only the entities belonging to the base.

Group/Post ...

Select Groups to Post:	**fem_base**
Apply	
Cancel	

Now, create the properties for the base.

Action:	**Create**
Dimension:	**2D**
Type:	**2D Solid**
Property Set Name:	**base**
Input Properties...	
Material Name:	**aluminum**
Thickness:	**1.0**
OK	
Select Members:	select all on screen
Add	
Apply	

Create a group containing all FEM and Geometry.

Group/Create ...

New Group Name:	**all**
Group Contents:	**Add All Entities**
Apply	
Cancel	

7. Create the Loads and Boundary Conditions for the model.

First, let’s fix the base in all degrees of freedom.

◆ **Loads/BCs**

Action:	**Create**
Object:	**Displacement**
Type:	**Nodal**
New Set Name:	**base_fixity**

Input Data...

Translations: **<0, 0, 0>**

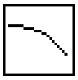
Rotations: **<0, 0, 0>**

OK

Select Application Region...

Geometry Filter: ◆ **Geometry**

In order to select the bottom edge of the base, use the following entity select icon:



Edge or Curve

Select Geometric Entities: select bottom edge of base

Add

OK

Apply

Now, create the displacement for pushing the block.

New Set Name: **block_push**

Input Data...

Translations: **<5, , >**

Rotations: **< , , >**

OK

Select Application Region...

Select Geometric Entities: select left edge of block

Add

OK

Apply

Create a BC to fix the block in the x-direction while the load is being applied.

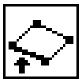
New Set Name: **horiz_hold**

Input Data...	
Translations:	**<0, , >**
Rotations:	**< , , >**
OK	
Select Geometric Entities:	select left edge of block
Add	
OK	
Apply	

Now, create a pressure on the top edge of the block (to simulate weight, load, or whatever - basically to keep the block pressed against the base).

Action:	**Create**
Object:	**Pressure**
Type:	**Element Uniform**
New Set Name:	**top_pressure**
Target Element Type:	**2D**
Input Data...	
Edge Pressure:	**100**
OK	
Select Application Region...	
Geometry Filter:	◆ **FEM**

Be sure to use the following entity select icon:



Edge of Element

Select 2D Elements or Edge:	select the top edges of the top elements of the block
Add	

OK

Apply

Create a contact definition for the block and the base.

Action:	**Create**
Object:	**Contact**
Type:	**Element Uniform**
Option:	**Deform-Deform**
New Set Name:	**interface**

Select Application Region...

Geometry Filter:	◆ **Geometry**
Master Surface:	**2D Solid Edge**
Slave Surface:	**2D Solid Edge**
Active Region:	**Master**
Select Surface Edges:	select left, bottom, and right edges of block

Add

Active Region:	**Slave**
Select Surface Edges:	select top edge of base

Add

OK

Input Data...

This is where you can add complications and customizations to the interface, such as friction.

Contact Type:	**General**
Sliding Type:	**Large**
Surface Behavior:	**Hard**
Friction Formulation:	**Penalty**
Viscous Damping:	**Off**

Penetration Type:	**One-Sided**

OK

Apply

8. Create two load cases for the analysis.

The first load case will consist of applying pressure to the top of the block, while the second will be a forced displacement with the pressure still on the block.

◆ **Load Cases**

Action:	**Create**
Load Case Name:	**press**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:	**Conta_interface** **Displ_base_fixity** **Displ_horiz_hold** **Press_top_pressure**

OK

Apply

Load Case Name:	**push**

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:	**Displ_block_push**
(select rows containing)	**Displ_horiz_hold**

Remove Selected Rows

OK

Apply

9. Submit the nonlinear analysis, creating two steps corresponding to the two load cases.

◆ **Analysis**

Action:	**Analyze**
Object:	**Entire Model**
Method:	**Full Run**
Step Creation...	
Job Step Name:	**step_1**
Solution Type:	**Nonlinear Static**
Select Load Cases...	
Available Load Cases:	**press**
OK	
Apply	
Job Step Name:	**step_2**
Solution Type:	**Nonlinear Static**
Solution Parameters...	
Max No of Increments:	**30**
OK	
Select Load Cases...	
Available Load Cases:	**push**
OK	
Apply	
Cancel	
Step Selection...	
Selected Job Steps:	**step_1** **step_2**
Apply	
Apply	

The non-linear analysis job **slide_block** will then be submitted for analysis to the workstation designated in the Submit Script (usually your local workstation).

The analysis job will take (on average) 5 to 10 minutes to run. When the job is done there will be a results file titled **slide_block.fil** in the same directory you started MSC/PATRAN in and the **slide_block.023** file will disappear.

You can monitor the progression of the job by looking at **slide_block.msg** and **slide_block.sta** files using the UNIX command **tail -lf [filename]**. You can also monitor the analysis in the background using the UNIX command **ps -a**.

10. Read in the results of the analysis.

◆ **Analysis**

Action: **Read Results**

Object: **Result Entities**

Method: **Translate**

Select Results File...

Available Files: **slide_block.fil**

OK

Apply

11. Display the results of the analysis.

First, post only the FEM of the model.

Group/Post ...

Select Groups to Post: **fem_all**

Apply

Cancel

Next, set the deformation scale factor to 1.0.

Display/Results ...

Scale Factor: **1.0**

◆ **Direct Multiplication**

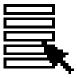
Apply

Cancel

Display the results at the end of the analysis, when the **Total Time=2.**

◆ **Results**

Click on the **Select Results** icon



Action: **Create**

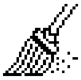
Object: **Quick Plot**

Select Result Cases: **Step2, Total time = 2**

Select Fringe Result: **Stress, Components**

Select Deformation Result: **Deformation, Displacements**

Now, let's animate the results and watch the process take place. First, clear the graphics by using the following icon:



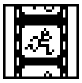
Reset Graphics

Next, deselect the fringe result **Stress, Components**.

Select Fringe Result:

■ **Animate**

Select the **Animation Options** icon



❐ **Animate Fringe**

Animation Method: ◆ **Ramped**

Number of Frames: **8**

OK

Apply

When done viewing, stop the animation and close the database.