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
Sliding Block

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

![Edge or Curve](image)

**Select Geometric Entities:** select bottom edge of base

Now, create the displacement for pushing the block.

**New Set Name:** block_push

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

**New Set Name:** horiz_hold
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

**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**
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
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

**Remove Selected Rows**

**OK**

**Apply**

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

◆ Analysis
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]
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.