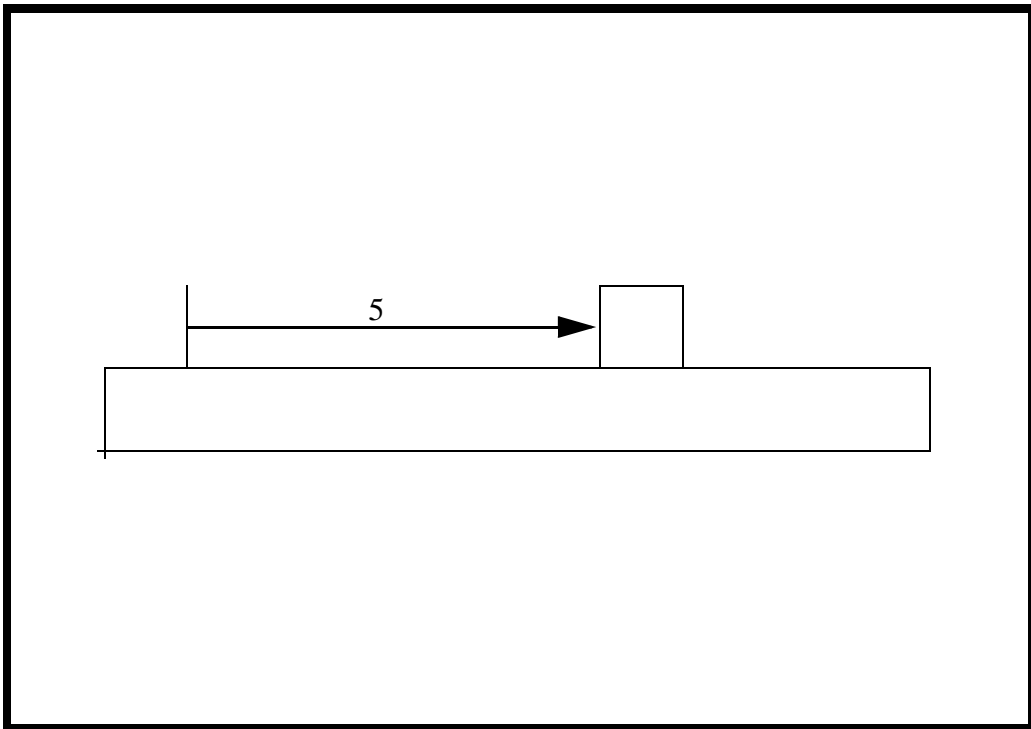

LESSON 26

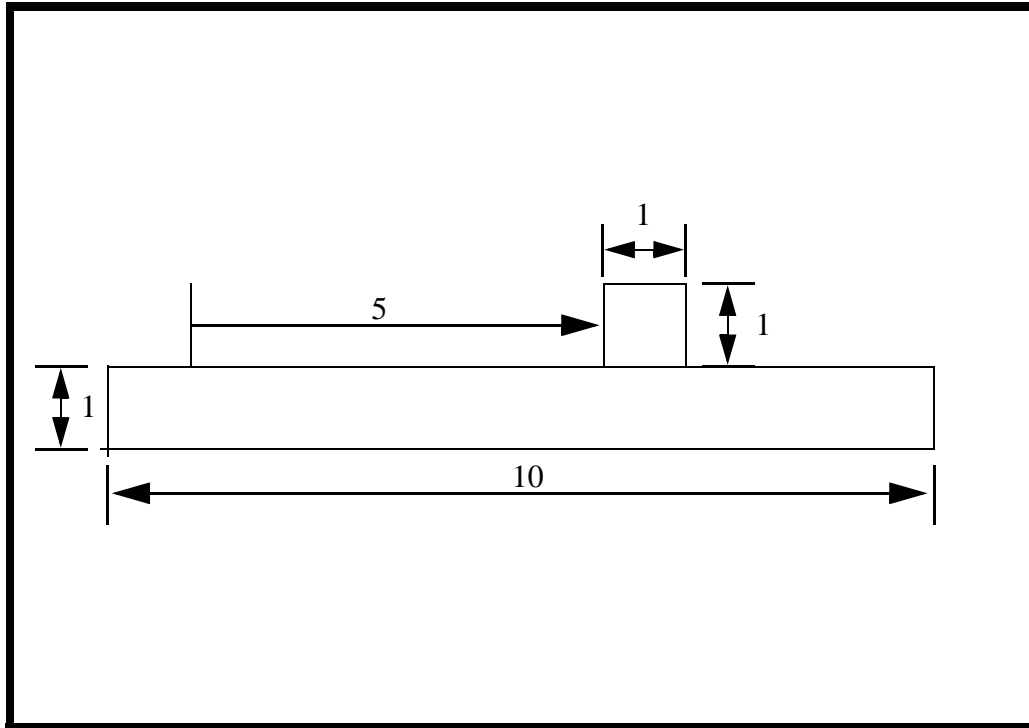
Sliding Block



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**.

File/New ...

New Database Name:

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.Marc**.

Tolerance: **Default**

Analysis Code:

Analysis Type:

2. Create the model geometry.

■ Geometry

Action:

Object:

Method:

Vector Coordinate List:

Vector Coordinate List:

Origin Coordinate List:

Group/Create ...

New Group Name:

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

■ **Elements**

Action:

Create

Object:

Mesh Seed

Method:

One Way Biased

Number:

4

L2 / L1 =

-2

Curve List:

select the bottom edge of base

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:

Mesher: **IsoMesh**

Surface List:

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:

Group Contents:

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:

Object:

Test:

Display Control: **Draw Normal Vectors**

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:

Object:

*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****OK****Apply**

Create the second material.

*Material Name:***aluminum**

Input Properties...

Elastic Modulus:

Poisson's Ratio:

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

First, lets deal with the block.

Group/Post ...

Select Groups to Post:

■ Properties

Action:

Dimension:

Type:

Property Set Name:

Input Properties...

Material Name:

Thickness:

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



Shell Element

Select Members:

Post only the entities belonging to the base.

Group/Post ...

Select Groups to Post:

Now, create the properties for the base.

Action:

Dimension:

Type:

Property Set Name:

Material Name:

Thickness:

Select Members:

Create a group containing all FEM and Geometry.

Group/Create ...

New Group Name:

Group Contents:

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

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

■ **Loads/BCs**

<i>Action:</i>	<input type="text" value="Create"/>
<i>Object:</i>	<input type="text" value="Displacement"/>
<i>Type:</i>	<input type="text" value="Nodal"/>
<i>New Set Name:</i>	<input type="text" value="base_fixity"/>

<i>Translations:</i>	<input type="text" value="<0, 0, 0>"/>
<i>Rotations:</i>	<input type="text" value="<0, 0, 0>"/>

Geometry Filter: ● **Geometry**

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



Curve or Edge

Select Geometric Entities:

Now, create the displacement for pushing the block.

New Set Name:

<i>Translations:</i>	<input type="text" value="<5, , >"/>
<i>Rotations:</i>	<input type="text" value="<, , >"/>

Select Geometric Entities:

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 Application Region...***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. Begin with the block.

Action:

Create

Object:

Contact

Type:

Element Uniform

Option:

Deformable Body

New Set Name:

interface_block

Target Element Type:

2D

Select Application Region...

Geometry Filter:

● **FEM**

Select Surfaces:

select block elements

Add

OK

Apply

Proceed with the same steps for the base.

Action:

Create

Object:

Contact

Type:

Element Uniform

Option:

Deformable Body

New Set Name:

Target Element Type:

Select Application Region...

Geometry Filter: FEM

Select Surfaces:

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:

Load Case Name:

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:

Load Case Name:

Assign/Prioritize Loads/BCs

Select LBCs to Add to Spreadsheet:

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

Load Step Creation...

Job Step Name:

step_1

Solution Type:

Static

Select Load Case...

Available Load Cases:

press

OK

Apply

Job Step Name:

step_2

Solution Type:

Static

Solution Parameters...

Load Increment Parameters...

Arclength Method:

Modified Riks/Ramm

Max # of Increments:

30

OK

OK

Select Load Case...

Available Load Cases:

push

OK

Apply

Cancel

Load Step Selection...*Selected Job Steps:*step_1
step_2

OK

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).

When the job is done there will be a results file titled **slide_block.t16** in the same directory you started MSC/PATRAN.

You can monitor the progression of the job by looking at *slide_block.log*, *slide_block.sts*, and *slid_block.out* files. You can also monitor the analysis using the **Monitor** option.

Action:

Monitor

Object:

Job

View Status File...

A successful analysis will result with a job exit number **3004**.

10. Read in the results of the analysis.

■ Analysis*Action:*

Read Results

Object:

Result Entities

Method:

Attach

Select Results File...*Available Files:*

slide_block.t16

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

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

■ **Results**

Action:

Create

Object:

Quick Plot

Select Result Cases:

step_2, A1:..., Time=2

Select Fringe Result:

Stress, Global System

Select Deformation Result:

Displacement, Translation

Apply

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, Global System**.

Select Fringe Result:

■ **Animate**

Select the **Animation Options** icon



Animate Fringe

Animation Method:

Ramped

Number of Frames:

8

Apply

When done viewing, stop the animation.

Close the database, and quit PATRAN.

This concludes this exercise.

