Created in COMSOL Multiphysics 6.2

Friction Between Contacting Rings

Introduction

This is a benchmark model involving stick-slip friction of a ring rolling inside another ring. The displacement of the inner ring is computed and compared to the analytical result (Ref. 1).

Model Definition

As illustrated in Figure 1, the geometry consists of two rings. The outer ring is 4 mm thick and has an inner radius of 156 mm. The inner ring has an inner radius of 100 mm and a thickness of 11.5 mm.

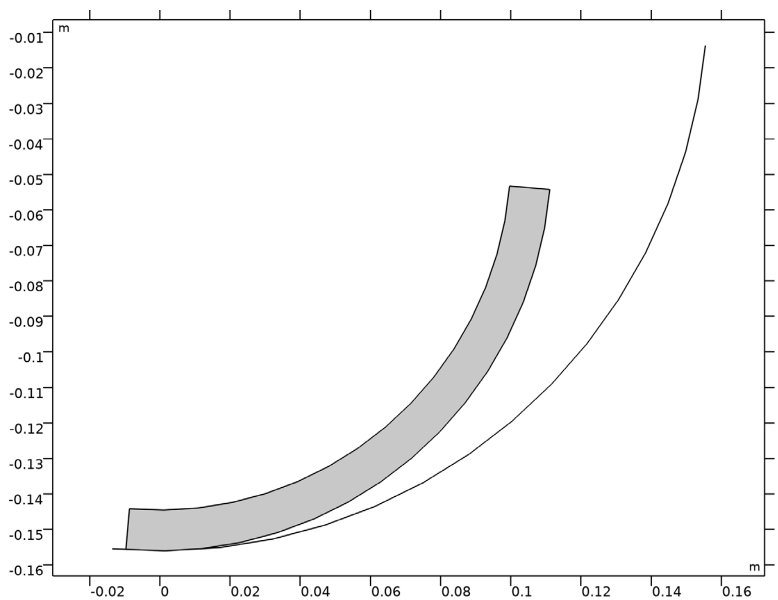

Figure 1: Model geometry.

The outer ring is fixed and rigid. Thus for the contact analysis only its mesh is required, without any physics attached. The inner ring is subjected to a prescribed rotation `phi` at its origin.

At the center of rotation, the resultant of the gravity load (P = 500 N) is applied to the inner ring.

A friction coefficient with the value 1 is used.

Results and Discussion

The analytical solution of the problem can be described as follows. The inner ring rolls along the outer ring until the tangential component of the gravity load becomes equal to the friction force (see Figure 2). At this critical point, slip occurs and the elevation of the inner ring reaches its maximum value.

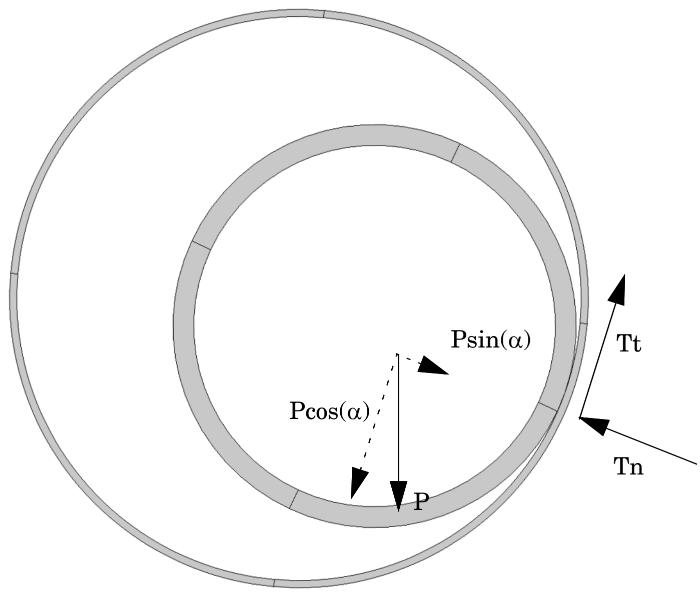

Figure 2: Representation of the contact and friction forces and the resultant of the gravity load.

The contact force corresponds to the normal component of the gravity load, $Tn = P\sin(\alpha)$. In this problem, the friction coefficient is 1, thus $Tn = Tt$ when sliding. As the critical position is reached when $Tt = P\cos(\alpha)$, the critical angle is $\alpha = 45^{\circ}$.

The maximum rolling distance is then $L = R \cdot \pi/4 = 122.5\ \mathrm{mm}$.

The vertical displacement of the center of the inner ring is defined as $Y = (R-r)(1-\cos(\alpha))$, where R is the inner radius of the outer ring and r is the outer radius of the inner ring. The maximum vertical displacement $Y_{max} = 13$ mm is reached at $\alpha = 45^{\circ}$.

[Figure 3](#) shows the von Mises stress distribution in the inner ring at the final step.

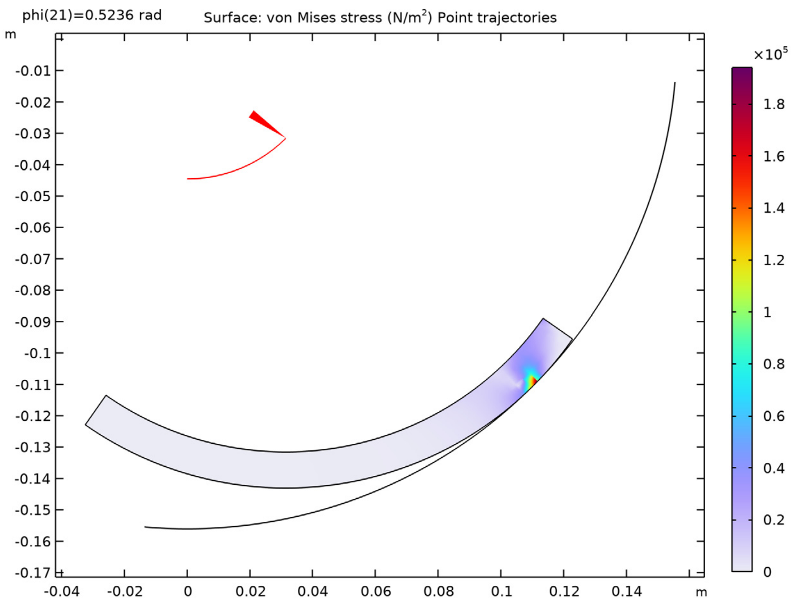

Figure 3: Stress distribution.

In Figure 4, you can see the elevation of the center of the inner ring with respect to its rotation angle. The computed maximum elevation is about 13 mm, and is in excellent agreement with the analytical solution.

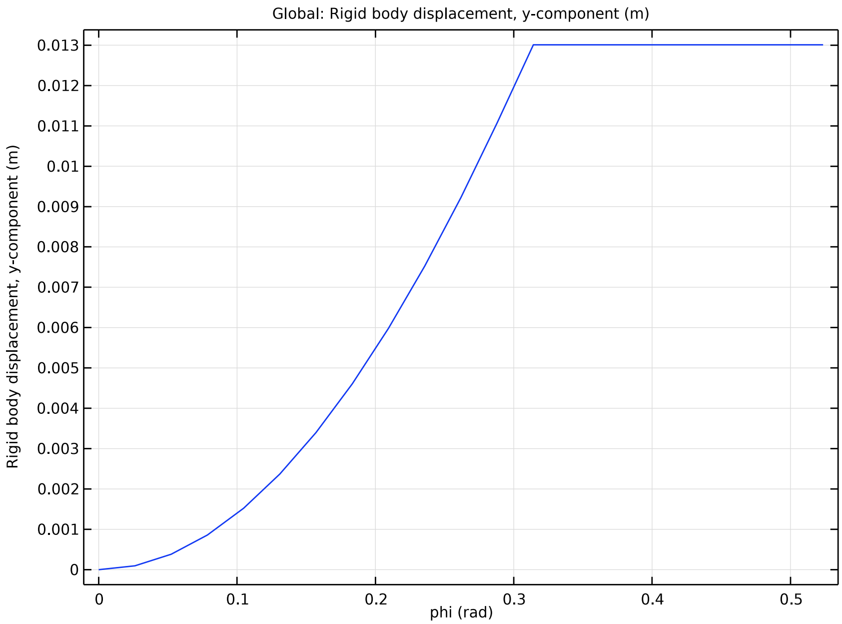

Figure 4: Elevation of the inner ring center versus applied rotation angle.

Figure 5 shows the contact pressure on the outer ring with respect to its length. The peak of the contact pressure occurs around 122 mm, as predicted by the analytical solution.

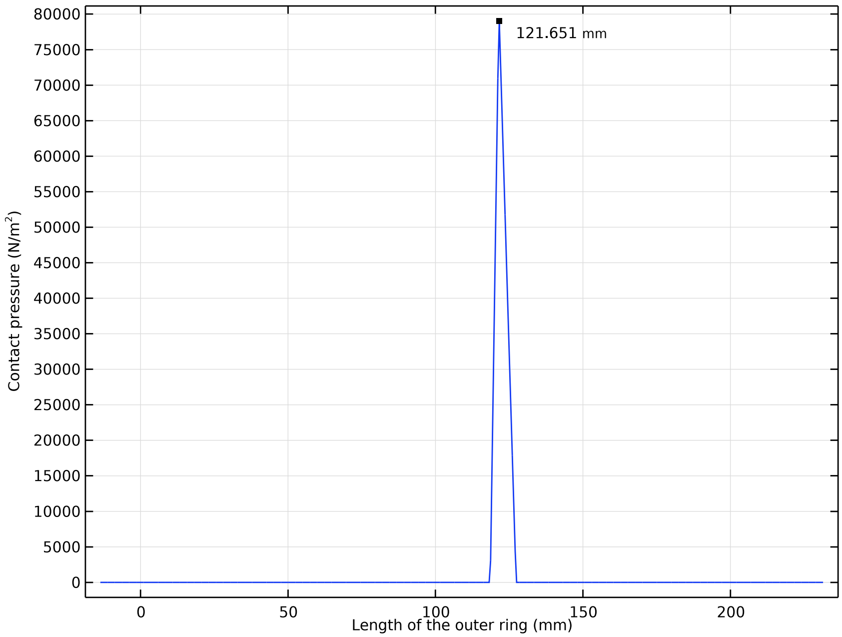

Figure 5: Contact pressure versus length of the outer ring.

Notes About the COMSOL Implementation

A rigid connector is used to prescribe the rotation of the inner ring, while leaving the translation free so that it can follow the curvature of the outer ring. The rigid connector is attached to the inner boundary of the inner ring.

Since the outer ring is assumed fixed and rigid, it requires no physics and it is sufficient to define a mesh on its the boundary.

To capture the transition between stick friction and slip friction, a small continuation parameter step is used. Furthermore, the augmented Lagrangian method is better suited for problems dominated by stick-slip friction than the default penalty method, and is thus used in this example.

The model is not stable in its initial configuration; there are possible rigid body displacements before contact is established. To avoid convergence problem, add a stabilization node to the contact feature which is automatically removed after few iterations.

Reference

1. Q. Feng and N.K. Prinja, "NAFEMS Benchmark Tests for Finite Element Modeling of Contact, Gapping and Sliding," *NAFEMS R0081*, 2001.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/contacting_rings`

Modeling Instructions

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **2D**.
2 In the **Select Physics** tree, select **Structural Mechanics>Solid Mechanics (solid)**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select Study** tree, select **General Studies>Stationary**.
6 Click **Done**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 In the table, enter the following settings:

Name	Expression	Value	Description
r1	160[mm]	0.16 m	Outer ring radius
r2	111.5[mm]	0.1115 m	Inner ring radius
t1	4[mm]	0.004 m	Outer ring thickness

Name	Expression	Value	Description
t2	11.5[mm]	0.0115 m	Inner ring thickness
y0	111.5[mm]-156[mm]	-0.0445 m	Inner ring center initial y-position

GEOMETRY 1

Circle 1 (c1)

1 In the **Geometry** toolbar, click **Circle**.

Only the inner surface of the outer ring needs to be modeled.

2 In the **Settings** window for **Circle**, locate the **Object Type** section.

3 From the **Type** list, choose **Curve**.

4 Locate the **Size and Shape** section. In the **Radius** text field, type `r1-t1`.

5 In the **Sector angle** text field, type `90`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-95`.

7 Click to expand the **Layers** section. Click **Build Selected**.

Circle 2 (c2)

1 In the **Geometry** toolbar, click **Circle**.

2 In the **Settings** window for **Circle**, locate the **Size and Shape** section.

3 In the **Radius** text field, type `r2`.

4 In the **Sector angle** text field, type `90`.

5 Locate the **Position** section. In the **y** text field, type `y0`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-95`.

7 Locate the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (m)
Layer 1	t2

8 Click **Build Selected**.

Delete Entities 1 (del1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Delete Entities**.

2 In the **Settings** window for **Delete Entities**, locate the **Entities or Objects to Delete** section.

3 From the **Geometric entity level** list, choose **Domain**.

4 On the object **c2**, select Domain 1 only.

Delete Entities 2 (del2)

1 Right-click **Geometry 1** and choose **Delete Entities**.

2 On the object **c1**, select Boundaries 2 and 3 only.

3 In the **Settings** window for **Delete Entities**, click **Build Selected**.

Form Union (fin)

1 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, locate the **Form Union/Assembly** section.

3 From the **Action** list, choose **Form an assembly**.

4 Click **Build Selected**.

DEFINITIONS

Variables 1

1 In the **Home** toolbar, click **Variables** and choose **Local Variables**.

2 In the **Settings** window for **Variables**, locate the **Geometric Entity Selection** section.

3 From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundary 1 only.

5 Locate the **Variables** section. In the table, enter the following settings:

Name	Expression	Unit	Description
L	(r1-t1)*(atan2(-y,-x)-pi/2)	m	Length of the outer ring

Contact Pair 1 (p1)

1 In the **Definitions** toolbar, click **Pairs** and choose **Contact Pair**.

2 In the **Settings** window for **Pair**, locate the **Source Boundaries** section.

3 Click **Paste Selection**.

4 In the **Paste Selection** dialog box, type 1 in the **Selection** text field.

5 Click **OK**.

6 In the **Settings** window for **Pair**, locate the **Destination Boundaries** section.

7 Click to select the **Activate Selection** toggle button.

8 Click **Paste Selection**.

9 In the **Paste Selection** dialog box, type 4 in the **Selection** text field.

10 Click **OK**.

MATERIALS

Material 1 (mat1)

1. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for **Material**, locate the **Material Contents** section.
3. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	210[GPa]	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	7850	kg/m³	Basic

SOLID MECHANICS (SOLID)

Contact 1

Use the **augmented Lagrange** method to evaluate the stick-slip contact.

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Contact 1**.
2. In the **Settings** window for **Contact**, locate the **Contact Method** section.
3. From the list, choose **Augmented Lagrangian**.
4. Locate the **Contact Pressure Penalty Factor** section. From the **Penalty factor control** list, choose **Manual tuning**.
5. In the f_p text field, type 1/50.

Friction 1

1. In the **Physics** toolbar, click **Attributes** and choose **Friction**.
2. In the **Settings** window for **Friction**, locate the **Friction Parameters** section.
3. In the μ text field, type 1.
4. Click the **Show More Options** button in the **Model Builder** toolbar.
5. In the **Show More Options** dialog box, in the tree, select the check box for the node **Physics>Advanced Physics Options**.
6. Click **OK**.
7. In the **Settings** window for **Friction**, click to expand the **Advanced** section.

8 Select the **Store accumulated slip** check box.

Contact 1

In the **Model Builder** window, click **Contact 1**.

Stabilization 1

In the **Physics** toolbar, click **Attributes** and choose **Stabilization**.

GLOBAL DEFINITIONS

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

2 In the **Settings** window for **Parameters**, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Value	Description
`phi`	`0[rad]`	0 rad	`Inner ring rotation angle`

SOLID MECHANICS (SOLID)

Rigid Connector 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Rigid Connector**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Rigid Connector**, locate the **Center of Rotation** section.

4 From the list, choose **User defined**.

5 Specify the $\mathbf{X}_c$ vector as

0	x
y0	y

6 Locate the **Prescribed Rotation** section. From the **By** list, choose **Prescribed rotation**.

7 In the ϕ_0 text field, type `-phi`.

Applied Force 1

1 In the **Physics** toolbar, click **Attributes** and choose **Applied Force**.

2 In the **Settings** window for **Applied Force**, locate the **Applied Force** section.

3 Specify the **F** vector as

0	x
-500	y

MESH 1

Mapped 1

In the **Mesh** toolbar, click **Mapped**.

Distribution 1

1. Right-click **Mapped 1** and choose **Distribution**.
2. Select Boundary 2 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `3`.

Distribution 2

1. In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.
2. Select Boundary 4 only.
3. In the **Settings** window for **Distribution**, locate the **Distribution** section.
4. In the **Number of elements** text field, type `60`.

Edge 1

1. In the **Mesh** toolbar, click **More Generators** and choose **Edge**.
2. Select Boundary 1 only.

Distribution 1

1. Right-click **Edge 1** and choose **Distribution**.
2. In the **Settings** window for **Distribution**, locate the **Distribution** section.
3. In the **Number of elements** text field, type `100`.
4. Click **Build All**.

STUDY 1

Step 1: Stationary

Set up an auxiliary continuation sweep for the `phi` parameter.

1. In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.
2. In the **Settings** window for **Stationary**, click to expand the **Results While Solving** section.

3 Select the **Plot** check box.

4 From the **Update at** list, choose **Steps taken by solver**.

5 Click to expand the **Study Extensions** section. Select the **Auxiliary sweep** check box.

6 Click **Add**.

7 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
phi (Inner ring rotation angle)	`range(0,pi/120,pi/6)`	`rad`

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** node, then click **Contact pressure (comp1.solid.Tn_p1)**.

4 In the **Settings** window for **Field**, locate the **Scaling** section.

5 In the **Scale** text field, type `1e5`.

6 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)>Dependent Variables 1** click **Friction force (spatial frame) (comp1.solid.Tt_p1)**.

7 In the **Settings** window for **Field**, locate the **Scaling** section.

8 In the **Scale** text field, type `1e5`.

9 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** node, then click **Parametric 1**.

10 In the **Settings** window for **Parametric**, click to expand the **Continuation** section.

11 Select the **Tuning of step size** check box.

12 In the **Initial step size** text field, type `pi/1000`.

13 In the **Maximum step size** text field, type `pi/1000`.

14 In the **Minimum step size** text field, type `pi/10000`.

15 In the **Model Builder** window, expand the **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1>Segregated 1** node, then click **Solid Mechanics**.

16 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

17 In the **Number of iterations** text field, type `15`.

18 In the **Model Builder** window, under **Study 1>Solver Configurations>Solution 1 (sol1)** right-click **Compile Equations: Stationary** and choose **Compute to Selected**.

RESULTS

Stress (solid)

Create a marker to make it easier to track the rotation of the inner ring. One way of doing it is to add an arrow to the default plot, which is generated below.

Point Trajectories 1

1 In the **Stress (solid)** toolbar, click **More Plots** and choose **Point Trajectories**.

2 In the **Settings** window for **Point Trajectories**, locate the **Trajectory Data** section.

3 In the **X-expression** text field, type `solid.u_rig1`.

4 In the **Y-expression** text field, type `y0+solid.v_rig1`.

5 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Color** list, choose **Red**.

6 Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

7 In the **Arrow, X-component** text field, type `cos(phi+5[deg])`.

8 In the **Arrow, Y-component** text field, type `sin(-phi-5[deg])`.

9 From the **Arrow type** list, choose **Cone**.

10 From the **Arrow base** list, choose **Head**.

STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Rigid Body Displacement

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Legend** section.

3 From the **Position** list, choose **Upper left**.

4 In the **Label** text field, type `Rigid Body Displacement`.

5 Locate the **Legend** section. Clear the **Show legends** check box.

Global 1

1 Right-click **Rigid Body Displacement** and choose **Global**.

2 In the **Settings** window for **Global**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Rigid connectors>Rigid Connector 1>Rigid body displacement (spatial frame) - m>solid.rig1.v - Rigid body displacement, y-component**.

3 In the **Rigid Body Displacement** toolbar, click **Plot**.

Contact Pressure, Outer Ring

1 In the **Home** toolbar, click **Add Plot Group** and choose **1D Plot Group**.

2 In the **Settings** window for **1D Plot Group**, locate the **Data** section.

3 From the **Parameter selection (phi)** list, choose **Last**.

4 In the **Label** text field, type Contact Pressure, Outer Ring.

5 Click to expand the **Title** section. From the **Title type** list, choose **None**.

6 Locate the **Plot Settings** section.

7 Select the **y-axis label** check box. In the associated text field, type Contact pressure (N/m²).

Line Graph 1

1 Right-click **Contact Pressure, Outer Ring** and choose **Line Graph**.

2 Select Boundary 1 only.

3 In the **Settings** window for **Line Graph**, locate the **y-Axis Data** section.

4 In the **Expression** text field, type dst2src_p1(solid.Tn_p1).

5 Locate the **x-Axis Data** section. From the **Parameter** list, choose **Expression**.

6 In the **Expression** text field, type L.

7 From the **Unit** list, choose **mm**.

Graph Marker 1

1 In the **Model Builder** window, right-click **Line Graph 1** and choose **Graph Marker**.

2 In the **Settings** window for **Graph Marker**, locate the **Display** section.

3 From the **Display** list, choose **Max**.

4 Locate the **Text Format** section. Select the **Show x-coordinate** check box.

5 Clear the **Show y-coordinate** check box.

6 Select the **Include unit** check box.

7 In the **Contact Pressure, Outer Ring** toolbar, click **Plot**.

Edge 2D 1

1 In the **Results** toolbar, click **More Datasets** and choose **Edge 2D**.

2 Select Boundary 4 only.

Parametric Extrusion 1D 1

In the **Results** toolbar, click **More Datasets** and choose **Parametric Extrusion 1D**.

Accumulated Slip

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Accumulated Slip` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Parametric Extrusion 1D 1**.

4 Click to expand the **Title** section. From the **Title type** list, choose **Label**.

Surface 1

1 Right-click **Accumulated Slip** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Expression** section.

3 In the **Expression** text field, type `gpeval(4,solid.sliptot)`.

Height Expression 1

1 Right-click **Surface 1** and choose **Height Expression**.

2 In the **Accumulated Slip** toolbar, click **Plot**.