Created in COMSOL Multiphysics 5.6

Sliding Wedge

Introduction

This is a benchmark model for contact and friction described in the NAFEMS publication in Ref. 1. An analytical solution exists, and this example includes a comparison of the COMSOL Multiphysics solution against the analytical solution.

Model Definition

A contactor wedge under the gravity load G is forced to slide due to a boundary load, F, over a target wedge surface, both infinitely thick (see Figure 1). Horizontal linear springs are also connected between the left vertical boundary of the contactor and the ground. The total spring stiffness is K.

This is a large sliding problem including contact pressure and friction forces. A boundary contact pair is created and the contact functionality in the Solid Mechanics interface is used to solve the contact problem. Both the penalty method and the augmented Lagrangian method are used, and friction is modeled with the Coulomb friction model. The augmented Lagrangian method is solved with both a segregated and a coupled solution method.

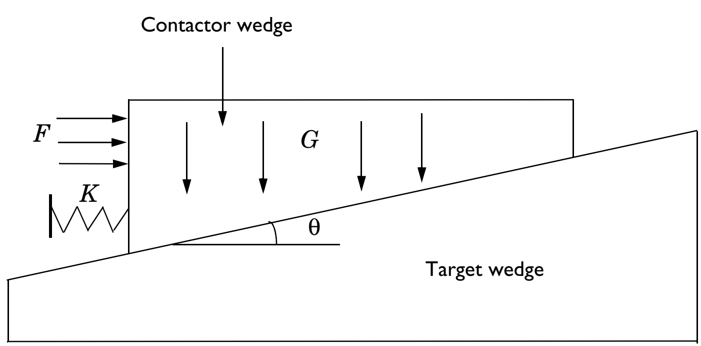

Figure 1: Sliding wedge with linear springs, a boundary load, and a gravity load.

The aim of this benchmark is to calculate the horizontal sliding distance and compare it with an elementary statics calculation. Three cases using different friction coefficients ($\mu = 0$; 0.1; 0.2) are analyzed.

For each friction coefficient, a specific total spring stiffness K is used (K = 1194 N/m; 882 N/m and 563.9 N/m respectively).

The horizontal applied force F = 1500 N, the total vertical gravity load G = 3058 N, the wedge angle is $\tan\theta = 0.1$.

For all study cases, the horizontal sliding distance is expected to be 1m.

The mesh is shown in Figure 2.

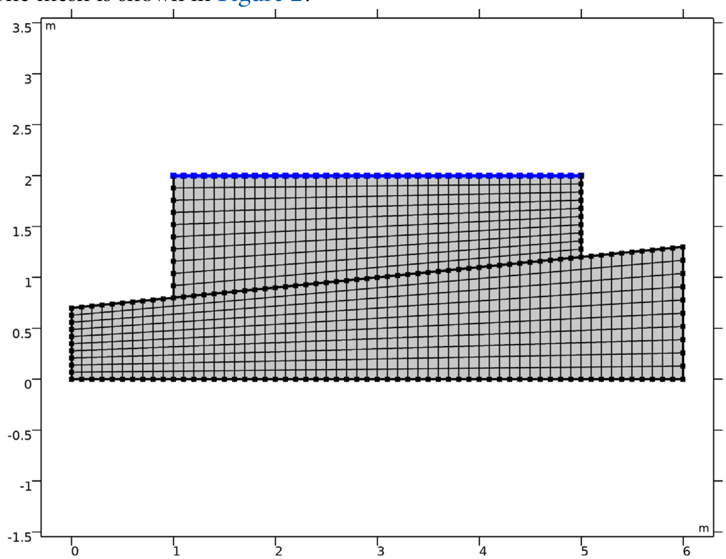

Figure 2: Quadrilateral elements are used to mesh the geometry.

The total number of elements in this model is 1000 and the number of degrees of freedom is 6484 for the displacement field.

Results and Discussion

The horizontal displacement computed for all friction cases agree very well with the reference data, see Ref. 1. For all cases, the difference is lower than 0.1%. Furthermore, all contact methods available in the Structural Mechanics Module converge to the same results. However, for this type of large sliding problem, the convergence and stability of the augmented Lagrangian method is superior to the penalty method.

Figure 3 below shows the result for the case $\mu = 0.2$, $K = 563.9$ N/m, and Figure 4 shows the contact pressure and friction forces for the same case. Both figures show the results obtained with the penalty method..

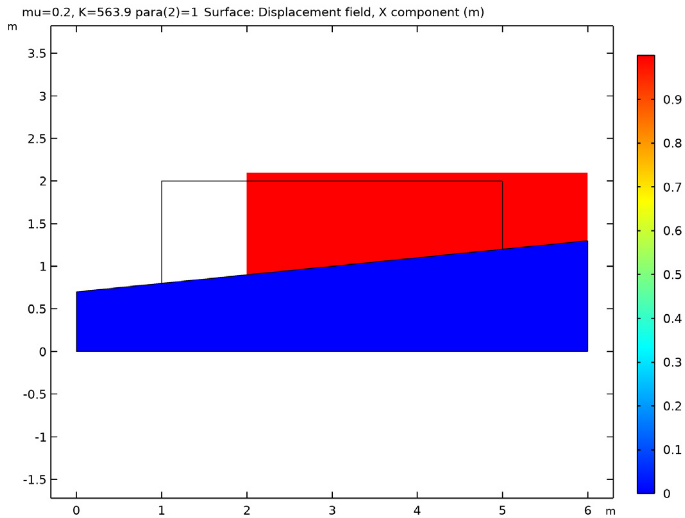

Figure 3: A surface plot of the x-displacement of the contactor wedge.

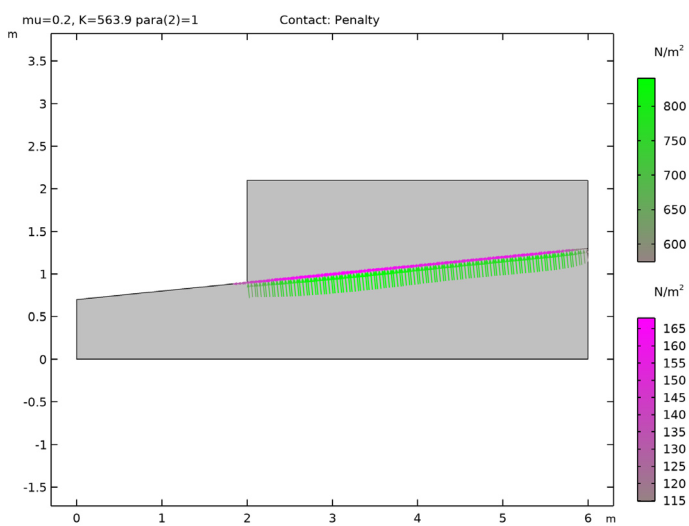

Figure 4: Contact pressure and friction forces acting on the contactor wedge.

Notes About the COMSOL Implementation

The initial unloaded state of the model is unstable and cause difficulties for the solver to find an initial solution. To avoid this issue, the first parameter step is set to 0.001. For this parameter value, a small amount of friction forces are present that stabilize the model.

The penalty method is not ideal for the type of large sliding problem with friction modeled in this example. While it in the limit will converge to the correct solution, the problem is stiff and ill-conditioned, meaning that small changes in the input can cause large changes to the results or even lead to no solution being found. In this example, the default solver suggestion does not give a stable solution, and the solver settings are modified to obtain a correct solution. Even with the modified settings, a warning from the linear solver gives an indication that the problem is ill-conditioned.

Reference

1. Feng Q., *NAFEMS Benchmark Tests for Finite Element Modelling of Contact, Gapping and Sliding*. NAFEMS Ref. R0081, UK, 2001.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/sliding_wedge`

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
G	3058[N]	3058 N	Gravity load
F	1500[N]	1500 N	Applied force
K	0[N/m]	0 N/m	Spring stiffness
mu	0	0	Friction coefficient
para	0	0	Computation parameter

GEOMETRY 1

Polygon 1 (pol1)

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

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

x (m)	y (m)
0	0
6	0
6	1.3
0	0.7

4 Click **Build All Objects**.

Rectangle 1 (r1)

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

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

3 In the **Width** text field, type 4.

4 In the **Height** text field, type 1.2.

5 Locate the **Position** section. In the **x** text field, type 1.

6 In the **y** text field, type 0.8.

Copy 1 (copy1)

1. In the **Geometry** toolbar, click **Transforms** and choose **Copy**.
2. Select the object **pol1** only.
3. In the **Settings** window for **Copy**, click **Build Selected**.

Difference 1 (dif1)

1. In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.
2. Select the object **r1** only.
3. In the **Settings** window for **Difference**, locate the **Difference** section.
4. Find the **Objects to subtract** subsection. Select the **Activate Selection** toggle button.
5. Select the object **copy1** only.
6. 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. From the **Pair type** list, choose **Contact pair**.
5. Click **Build Selected**.
6. Click the **Zoom Extents** button in the **Graphics** toolbar.

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	206[GPa]	Pa	Basic
Poisson's ratio	nu	0.3	1	Basic
Density	rho	6000[kg/m^3]	kg/m³	Basic

SOLID MECHANICS (SOLID)

Body Load 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** and choose **Volume Forces>Body Load**.

2 Select Domain 2 only.

3 In the **Settings** window for **Body Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

0	x
-G*para	y

Contact 1

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

2 In the **Settings** window for **Contact**, locate the **Pair Selection** section.

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (ap1)** in the **Pairs** list.

5 Click **OK**.

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 `mu`.

4 Locate the **Initial Value** section. From the **Previous contact state** list, choose **In contact**.

Spring Foundation 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Spring Foundation**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Spring Foundation**, locate the **Spring** section.

4 From the **Spring type** list, choose **Total spring constant**.

5 From the list, choose **Diagonal**.

6 In the $\mathbf{k}_{tot}$ table, enter the following settings:

K	0
0	0

Boundary Load 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Load**.

2 Select Boundary 5 only.

3 In the **Settings** window for **Boundary Load**, locate the **Force** section.

4 From the **Load type** list, choose **Total force**.

5 Specify the $\mathbf{F}_{tot}$ vector as

F*para	x
0	y

Fixed Constraint 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Fixed Constraint**.

2 Select Boundary 2 only.

MESH 1

Mapped 1

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

Distribution 1

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundaries 1 and 5 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 10.

Distribution 2

1 In the **Model Builder** window, 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 60.

Distribution 3

1 Right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 7 only.

3 In the **Settings** window for **Distribution**, locate the **Distribution** section.

4 In the **Number of elements** text field, type 40.

5 Click **Build All**.

STUDY 1

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
mu (Friction coefficient)	0 0.1 0.2	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
K (Spring stiffness)	1194 882 563.9	N/m

Step 1: Stationary

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

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, click to expand the **Study Extensions** section.

3 Select the **Auxiliary sweep** check box.

4 Click **Add**.

5 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Computation parameter)	1e-3 1	

Set a stricter tolerance and tune the parameter stepping of the auxiliary sweep to improve the convergence of the model. The convergence is also improved by changing the nonlinear solver to Constant Newton.

Solution 1 (sol1)

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

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

3 In the **Settings** window for **Stationary Solver**, locate the **General** section.

4 In the **Relative tolerance** text field, type `1e-6`.

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

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

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

8 In the **Initial step size** text field, type `1e-2`.

9 In the **Minimum step size** text field, type `1e-6`.

10 From the **Predictor** list, choose **Linear**.

11 In the **Model Builder** window, click **Fully Coupled 1**.

12 In the **Settings** window for **Fully Coupled**, click to expand the **Method and Termination** section.

13 From the **Nonlinear method** list, choose **Constant (Newton)**.

14 In the **Model Builder** window, click **Study 1**.

15 In the **Settings** window for **Study**, type `Study 1: Penalty` in the **Label** text field.

16 In the **Study** toolbar, click **Compute**.

RESULTS

Displacement: Penalty

In the **Settings** window for **2D Plot Group**, type `Displacement: Penalty` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Displacement: Penalty** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics> Displacement>Displacement field - m>u - Displacement field, X component**.

3 In the **Displacement: Penalty** toolbar, click **Plot**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Applied Loads: Penalty

1 In the **Model Builder** window, under **Results** click **Applied Loads (solid)**.

2 In the **Settings** window for **Group**, type `Applied Loads: Penalty` in the **Label** text field.

Contact: Penalty

1 In the **Model Builder** window, under **Results** click **Contact Forces (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Contact: Penalty` in the **Label** text field.

Gray Surfaces

1 In the **Model Builder** window, expand the **Contact: Penalty** node, then click **Gray Surfaces**.

2 In the **Contact: Penalty** toolbar, click **Plot**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

Follow the instructions below to evaluate the horizontal displacement for all three friction cases.

Point Evaluation: Penalty

1 In the **Results** toolbar, click 8.85 e-12 **Point Evaluation**.

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation: Penalty` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 1: Penalty/ Parametric Solutions 1 (sol2)**.

4 From the **Parameter selection (para)** list, choose **Last**.

5 From the **Table columns** list, choose **mu, K**.

6 Select Point 8 only.

7 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X component**.

8 Click **Evaluate**.

Now, solve the model using the augmented Lagrangian formulation. Explore both a segregated and a coupled solution method.

SOLID MECHANICS (SOLID)

Contact 2

1 In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Contact 1** and choose **Duplicate**.

2 In the **Settings** window for **Contact**, locate the **Contact Method** section.

3 From the **Formulation** list, choose **Augmented Lagrangian**.

Contact 3

1 Right-click **Contact 2** and choose **Duplicate**.

2 In the **Settings** window for **Contact**, locate the **Contact Method** section.

3 From the **Solution method** list, choose **Fully coupled**.

ADD STUDY

1 In the **Home** toolbar, click **Add Study** to open the **Add Study** window.

2 Go to the **Add Study** window.

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies>Stationary**.

4 Click **Add Study**.

5 In the **Home** toolbar, click **Add Study** to close the **Add Study** window.

STUDY 2

Parametric Sweep

1 In the **Study** toolbar, click **Parametric Sweep**.

2 In the **Settings** window for **Parametric Sweep**, locate the **Study Settings** section.

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
mu (Friction coefficient)	0 0.1 0.2	

5 Click **Add**.

6 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
K (Spring stiffness)	1194 882 563.9	N/m

Step 1: Stationary

1 In the **Model Builder** window, click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 1**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 3**.

7 Click **Disable**.

8 Locate the **Study Extensions** section. Select the **Auxiliary sweep** check box.

9 Click **Add**.

10 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
para (Computation parameter)	`1e-3 1`	

11 In the **Model Builder** window, click **Study 2**.

12 In the **Settings** window for **Study**, type `Study 2: Augmented Lagrangian, Segregated` in the **Label** text field.

In this example the contact forces are very small, so it is necessary so set proper scales for these variables.

Solution 6 (sol6)

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

2 In the **Model Builder** window, expand the **Solution 6 (sol6)** node.

3 In the **Model Builder** window, expand the **Study 2: Augmented Lagrangian, Segregated>Solver Configurations>Solution 6 (sol6)>Dependent Variables 1** node, then click **Friction force (spatial frame) (comp1.solid.Tt_ap1)**.

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

5 In the **Scale** text field, type `100`.

6 In the **Model Builder** window, click **Contact pressure (comp1.solid.Tn_ap1)**.

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

8 In the **Scale** text field, type `1000`.

9 In the **Model Builder** window, expand the **Study 2: Augmented Lagrangian, Segregated>Solver Configurations>Solution 6 (sol6)>Stationary Solver 1** node, then click **Parametric 1**.

10 In the **Settings** window for **Parametric**, locate the **Continuation** section.

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

12 In the **Initial step size** text field, type `0.1`.

13 In the **Maximum step size** text field, type `1`.

14 In the **Study** toolbar, click **Compute**.

Similarly, add a third study for the augmented Lagrangian formulation with a coupled solution method and compute the solution. Disable **Contact 1** and **Contact 2**, and use a Constant (Newton) solver.

RESULTS

Displacement: Augmented Lagrangian, Segregated

In the **Settings** window for **2D Plot Group**, type `Displacement: Augmented Lagrangian, Segregated` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Displacement: Augmented Lagrangian, Segregated** node, then click **Surface 1**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)>Solid Mechanics>Displacement>Displacement field - m>u - Displacement field, X component**.

3 In the **Displacement: Augmented Lagrangian, Segregated** toolbar, click **Plot**.

4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Applied Loads: Augmented Lagrangian, Segregated

1 In the **Model Builder** window, under **Results** click **Applied Loads (solid)**.

2 In the **Settings** window for **Group**, type `Applied Loads: Augmented Lagrangian, Segregated` in the **Label** text field.

Contact: Augmented Lagrangian, Segregated

1 In the **Model Builder** window, under **Results** click **Contact Forces (solid)**.

2 In the **Settings** window for **2D Plot Group**, type `Contact: Augmented Lagrangian, Segregated` in the **Label** text field.

Gray Surfaces

1 In the **Model Builder** window, expand the **Contact: Augmented Lagrangian, Segregated** node, then click **Gray Surfaces**.

2 In the **Contact: Augmented Lagrangian, Segregated** toolbar, click **Plot**.

3 Click the **Zoom Extents** button in the **Graphics** toolbar.

Point Evaluation: Augmented Lagrangian, Segregated

1 In the **Model Builder** window, right-click **Point Evaluation: Penalty** and choose **Duplicate**.

2 In the **Settings** window for **Point Evaluation**, type `Point Evaluation: Augmented Lagrangian, Segregated` in the **Label** text field.

3 Locate the **Data** section. From the **Dataset** list, choose **Study 2: Augmented Lagrangian, Segregated/Parametric Solutions 2 (sol7)**.

4 Click ▾ next to **Evaluate**, then choose **New Table**.

Repeat the same steps for the datasets and plots generated by **Study 3: Augmented Lagrangian, Coupled**.

Prepare the model for later use by disabling the second and third contact nodes in **Study 1: Penalty**.

STUDY 1: PENALTY

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1: Penalty** click **Step 1: Stationary**.

2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.

3 Select the **Modify model configuration for study step** check box.

4 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid), Controls spatial frame>Contact 2**.

5 Click **Disable**.

6 In the **Physics and variables selection** tree, select **Component 1 (comp1)> Solid Mechanics (solid), Controls spatial frame>Contact 3**.

7 Click **Disable**.