



# Sliding Wedge

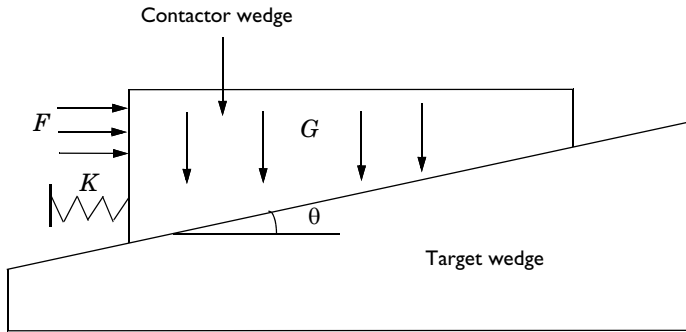
## Introduction

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. The penalty method, the two formulations of the augmented Lagrangian method, and the Nitsche method are used, and friction is modeled with the Coulomb friction model.



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

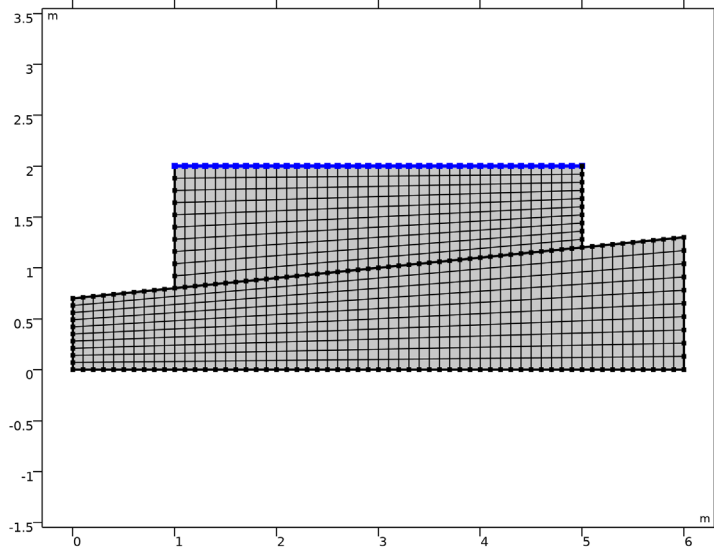
The aim of this benchmark is to calculate the horizontal sliding distance and compare it with an elementary statics calculation. Three cases using different friction coefficients ( $\mu = 0$ ; 0.1; 0.2) are analyzed.

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

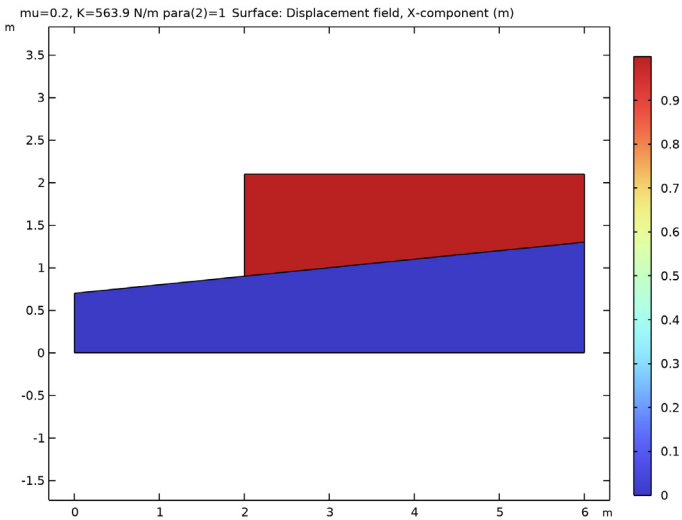


Figure 3: A surface plot of the x-displacement of the contactor wedge.

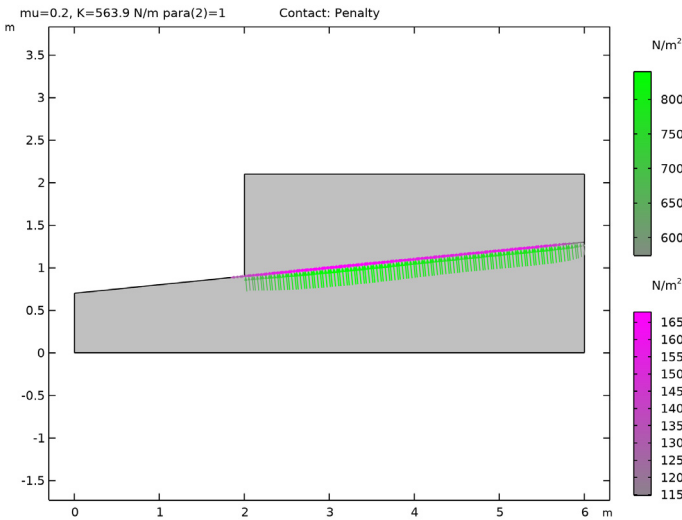


Figure 4: Contact pressure and friction forces acting on the contactor wedge.

## *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.

Neither the penalty method nor the Nitsche method is ideal for the type of large sliding problem with friction modeled in this example. While it in the limit the problem will converge to the correct solution, it 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 I

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


### Polygon I (pol)

- 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 I (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 Click to select the  **Activate Selection** toggle button for **Objects to subtract**.
- 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	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.3	1	Young's modulus and Poisson's ratio
Density	rho	6000 [kg/m^3]	kg/m <sup>3</sup>	Basic

**SOLID MECHANICS (SOLID)**


*Contact I*

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

*Friction I*


- 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  $\mu$  text field, type  $\mu$ .

*Body Load I*

- 1 In the **Physics** toolbar, click  **Domains** and choose **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  $\mathbf{F}_{\text{tot}}$  vector as


0	x
-G*para	y

*Spring Foundation I*

- 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  $\mathbf{k}_{\text{tot}}$  table, enter the following settings:

K	0
0	0

*Boundary Load I*


- 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  $\mathbf{F}_{\text{tot}}$  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, under **Study 1>Solver Configurations>Solution 1 (sol1)>Stationary Solver 1** 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 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 4 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.
- 5 Click **OK**.
- 6 In the **Displacement: Penalty** toolbar, click  **Plot**.
- 7 Click the  **Zoom Extents** button in the **Graphics** toolbar.

## ADD PREDEFINED PLOT

- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.



- 3 In the tree, select **Study 1: Penalty/Parametric Solutions 1 (sol2)>Solid Mechanics>Contact Forces (solid)** and **Study 1: Penalty/Parametric Solutions 1 (sol2)>Solid Mechanics>Applied Loads (solid)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

## RESULTS

### *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  **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 1

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



### Contact 2

- 1 In the **Model Builder** window, click **Contact 2**.
- 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**.
- 6 In the **Settings** window for **Contact**, locate the **Contact Method** section.
- 7 From the list, choose **Augmented Lagrangian**.
- 8 Right-click **Contact 2** and choose **Duplicate**.

### Contact 3



- 1 In the **Model Builder** window, click **Contact 3**.
- 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 Right-click and choose **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 tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 3**.

5 Right-click and choose **Disable**.

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

7 Click  **Add**.

8 In the table, enter the following settings:

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

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

10 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, under **Study 2: Augmented Lagrangian, Segregated> Solver Configurations>Solution 6 (sol6)>Dependent Variables 1** 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 Locate the **Coloring and Style** section. Click  **Change Color Table**.
- 4 In the **Color Table** dialog box, select **Rainbow>RainbowLight** in the tree.
- 5 Click **OK**.
- 6 In the **Displacement: Augmented Lagrangian, Segregated** toolbar, click  **Plot**.
- 7 Click the  **Zoom Extents** button in the **Graphics** toolbar.

## ADD PREDEFINED PLOT



- 1 In the **Home** toolbar, click  **Add Predefined Plot** to open the **Add Predefined Plot** window.
- 2 Go to the **Add Predefined Plot** window.
- 3 In the tree, select **Study 2: Augmented Lagrangian, Segregated/Parametric Solutions 2 (sol7)>Solid Mechanics>Contact Forces (solid)**.
- 4 Click **Add Plot** in the window toolbar.
- 5 In the **Home** toolbar, click  **Add Predefined Plot** to close the **Add Predefined Plot** window.

## RESULTS

### *Contact: Augmented Lagrangian, Segregated*

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: Penalty*

In the **Model Builder** window, under **Results>Derived Values** right-click **Point Evaluation: Penalty** and choose **Duplicate**.

### *Point Evaluation: Augmented Lagrangian, Segregated*

- 1 In the **Model Builder** window, under **Results>Derived Values** click **Point Evaluation: Penalty I**.
- 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**.



Solve the model using the Nitsche method. Use the same solver settings as in **Study 1: Penalty**

**SOLID MECHANICS (SOLID)**



*Contact 3*

In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** right-click **Contact 3** and choose **Duplicate**.

*Contact 4*



- 1 In the **Model Builder** window, click **Contact 4**.
- 2 In the **Settings** window for **Contact**, locate the **Contact Method** section.
- 3 From the list, choose **Nitsche**.

**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 Right-click and choose **Add Study**.
- 5 In the **Home** toolbar, click  **Add Study** to close the **Add Study** window.

**STUDY 4**

*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 **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	

### Solution 16 (sol16)

- 1 In the **Study** toolbar, click  **Show Default Solver**.
- 2 In the **Model Builder** window, expand the **Solution 16 (sol16)** 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 4>Solver Configurations>Solution 16 (sol16)>Stationary Solver 1** node, then click **Parametric 1**.
- 6 In the **Settings** window for **Parametric**, locate 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, under **Study 4>Solver Configurations>Solution 16 (sol16)>Stationary Solver 1** click **Fully Coupled 1**.
- 12 In the **Settings** window for **Fully Coupled**, locate the **Method and Termination** section.
- 13 From the **Nonlinear method** list, choose **Constant (Newton)**.
- 14 In the **Model Builder** window, click **Study 4**.
- 15 In the **Settings** window for **Study**, type Study 4: Nitsche in the **Label** text field.
- 16 In the **Study** toolbar, click  **Compute**.

## RESULTS

### Stress (solid)

Repeat the same steps for the datasets and plots generated by **Study 4: Nitsche**.

Prepare the model for later use by making sure that the correct **Contact** node is active in all previous studies.

## **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 tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 2**.
- 5 Right-click and choose **Disable**.
- 6 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 3**.
- 7 Right-click and choose **Disable**.
- 8 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 4**.
- 9 Right-click and choose **Disable**.

## **STUDY 2: AUGMENTED LAGRANGIAN, SEGREGATED**

- 1 In the **Model Builder** window, under **Study 2: Augmented Lagrangian, Segregated** click **Step 1: Stationary**.
- 2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
- 3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 4**.
- 4 Right-click and choose **Disable**.

## **STUDY 3: AUGMENTED LAGRANGIAN, COUPLED**

- 1 In the **Model Builder** window, under **Study 3: Augmented Lagrangian, Coupled** 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 tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 4**.
- 5 Right-click and choose **Disable**.

