Created in COMSOL Multiphysics 6.2

Cylinder Roller Contact

Introduction

Consider an infinitely long steel cylinder resting on a flat aluminum foundation, where both structures are elastic. The cylinder is subjected to a point load along its top. The objective of this study is to find the contact pressure distribution and the length of contact between the foundation and the cylinder. An analytical solution exists, and this tutorial includes a comparison with the COMSOL Multiphysics solution. The application is based on a NAFEMS benchmark (see Ref. 1).

Model Definition

This is a plane strain problem and the 2D Solid Mechanics interface from the Structural Mechanics Module is thus suitable. The 2D geometry is further cut in half at the vertical symmetry axis.

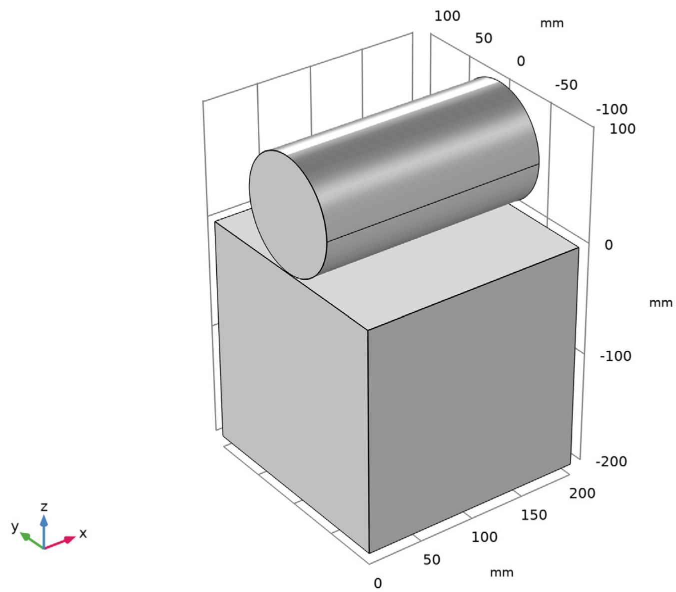


Figure 1: Model geometry.

In 2D, the cylinder is subjected to a point load along its top with an intensity of 35 kN/mm. Both the cylinder and block material are elastic, homogeneous, and isotropic.

The contact modeling in this example only includes the frictionless part of the example described in Ref. 1. The problem is implemented with the Solid Mechanics interface, and four studies are set up to compare the default penalty contact method, the two formulations of the augmented Lagrangian method, and Nitsche method.

Results and Discussion

Figure 2 depicts the deformed shape and the von Mises stress distribution obtained with the penalty contact method.

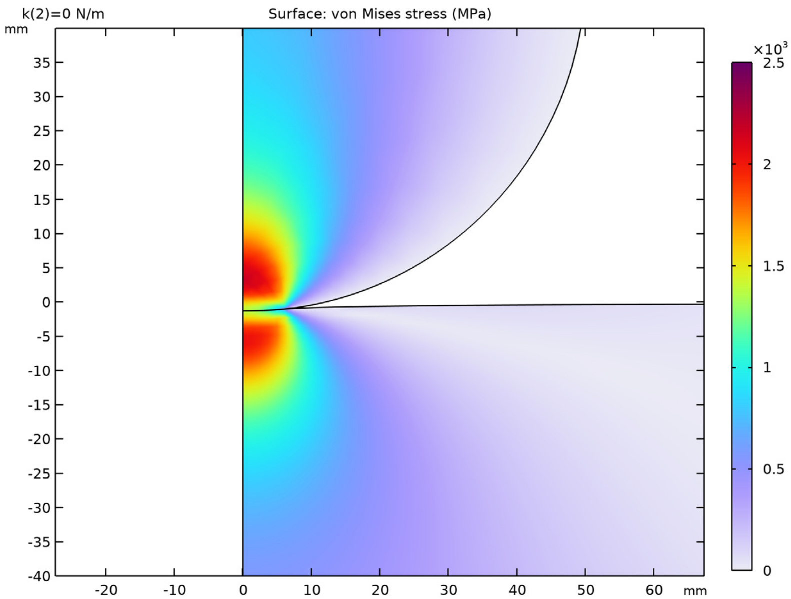


Figure 2: Deformation and von Mises stress at the contact area.

The analytical solution for the contact pressure as a function of the *x*-coordinate is

$$P = \sqrt{\frac{F_n E'}{2\pi R'} \times \left(1 - \left(\frac{x}{a}\right)^2\right)}$$

$$a = \sqrt{\frac{8F_n R'}{\pi E'}}$$

where F_n is the applied load per unit length, E' is the combined elasticity modulus, and R' is the combined radius. The combined Young's modulus and radius are defined as:

$$E' = \frac{2E_1E_2}{E_2(1-\nu_1^2)+E_1(1-\nu_2^2)}$$

$$R' = \lim_{R_2 \to \infty} \frac{R_1R_2}{R_1+R_2} = R_1$$

In these equations, E_1 and E_2 are Young's modulus of the roller and the block, respectively, and R_1 is the radius of the roller. Combining these equations results in a contact length of 6.21 mm and a maximum contact pressure of 3585 MPa.

Figure 3 depicts the contact pressure along the contact area for both the analytical and the four COMSOL Multiphysics solutions.

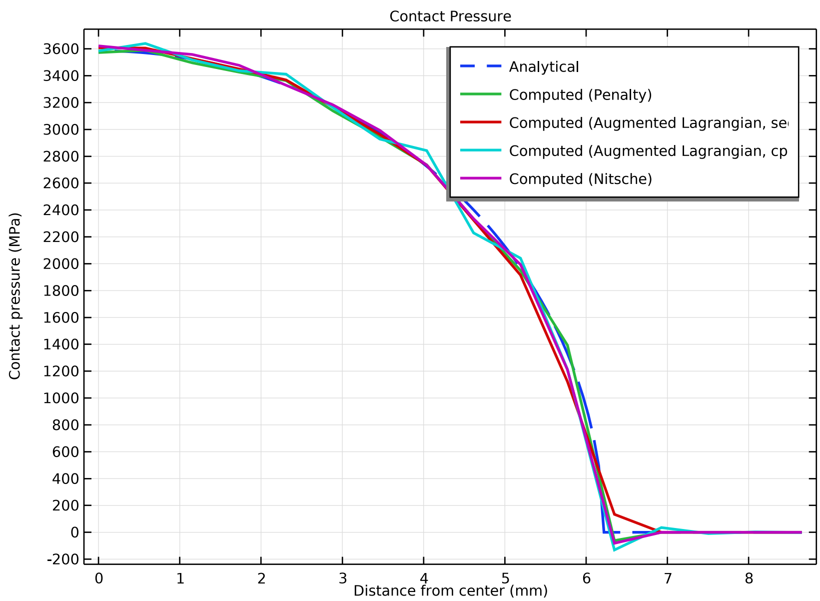


Figure 3: Analytical pressure distribution (dashed line) and COMSOL Multiphysics solutions (solid lines).

Notes About the COMSOL Implementation

The Structural Mechanics Module supports contact boundary conditions using contact pairs. The contact pair is defined by a source boundary and a destination boundary. The destination boundary is the one which is coupled to the source boundary if contact is established. The terms source and destination should be interpreted as in "the destination receives its displacements from the source." As a result, the contact pressure variable is available on the destination boundary. The mesh on the destination side should always be finer than on the source side.

In this example, the contact boundary pair consists of a flat source boundary and a curved destination boundary.

The cylinder is initially stabilized with a weak spring. A good approximation of the spring coefficient is to use the value of the external pressure — in this case the external point load divided by one fifth of the initial gap. In a second step, the spring is removed to arrive at the final solution.

The small size of the contact region necessitates a local mesh refinement. Use an unstructured mesh for the cylindrical domain and a mapped mesh for the aluminum block. The block geometry requires some modification to set up a refined mesh area.

References

1. A.W.A. Konter, *Advanced Finite Element Contact Benchmarks*, NAFEMS, 2006.

2. M.A. Crisfield, *Non-linear Finite Element Analysis of Solids and Structures, volume 2: Advanced Topics*, John Wiley & Sons, London, 1997.

Application Library path: `Structural_Mechanics_Module/Verification_Examples/cylinder_roller_contact`

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 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `cylinder_roller_contact.txt`.

DEFINITIONS

Variables 1

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

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

3 In the table, enter the following settings:

Name	Expression	Unit	Description
p_analytical	pmax*sqrt(1-(x/a)^2)	N/m²	Analytical contact pressure

GEOMETRY 1

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

2 In the **Settings** window for **Geometry**, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Now create the geometry. Recall that you only need to model one half of the 2D cross section.

Circle 1 (c1)

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 `R`.

4 In the **Sector angle** text field, type `180`.

5 Locate the **Position** section. In the **y** text field, type `R+dist`.

6 Locate the **Rotation Angle** section. In the **Rotation** text field, type `-90`.

7 Click **Build Selected**.

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 `d/2`.

4 In the **Height** text field, type `d`.

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

6 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
Layer 1	d/2

7 Click **Build Selected**.

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

Square 1 (sq1)

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

2 In the **Settings** window for **Square**, locate the **Size** section.

3 In the **Side length** text field, type `R/2`.

4 Locate the **Position** section. In the **y** text field, type `-R/2`.

5 Click **Build Selected**.

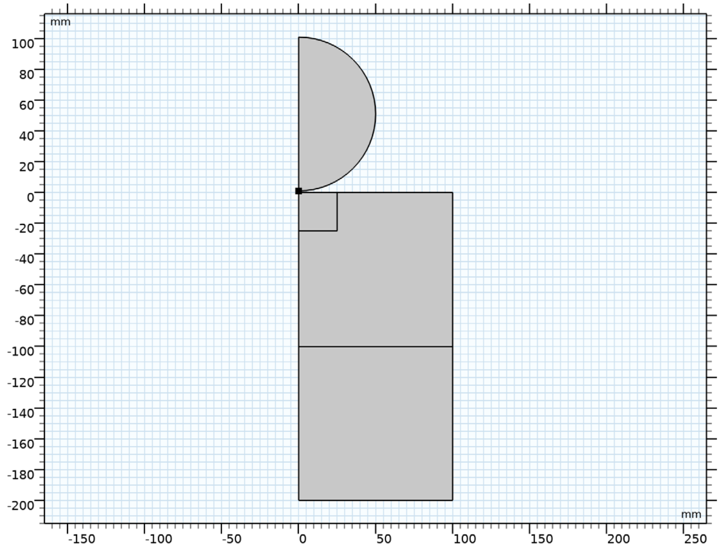
Point 1 (pt1)

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

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

3 In the **y** text field, type `dist`.

4 Click **Build Selected**.



Rotate 1 (rot1)

1 In the **Geometry** toolbar, click **Transforms** and choose **Rotate**.

2 Select the object **pt1** only.

3 In the **Settings** window for **Rotate**, locate the **Rotation** section.

4 In the **Angle** text field, type `10`.

5 Locate the **Center of Rotation** section. In the **y** text field, type `R+dist`.

6 Click **Build Selected**.

Convert to Solid 1 (csol1)

1 In the **Geometry** toolbar, click **Conversions** and choose **Convert to Solid**.

2 Click in the **Graphics** window and then press Ctrl+A to select all objects.

3 In the **Settings** window for **Convert to Solid**, 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 Clear the **Create pairs** check box.

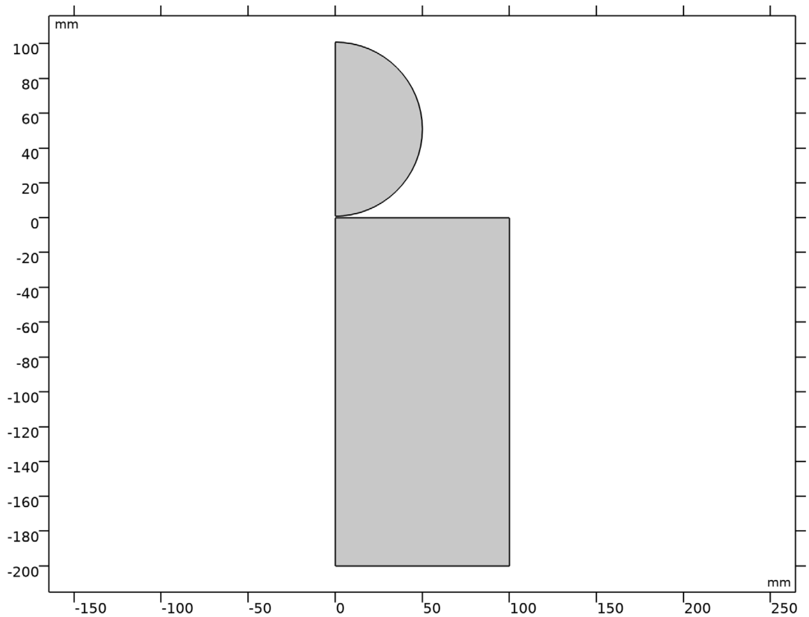
Mesh Control Domains 1 (mcd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Mesh Control Domains**.

2 On the object **fin**, select Domains 1–3 only.

3 In the **Geometry** toolbar, click **Build All**.

The model geometry is now complete.



DEFINITIONS

Contact Pair 1 (p1)

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

2 Select Boundary 3 only.

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

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

5 Select Boundary 7 only.

SOLID MECHANICS (SOLID)

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

2 In the **Settings** window for **Solid Mechanics**, locate the **Thickness** section.

3 In the *d* text field, type `th`.

Symmetry 1

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

2 Select Boundaries 1, 4, and 5 only.

Fixed Constraint 1

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

2 Select Boundary 2 only.

Point Load 1

1 In the **Physics** toolbar, click **Points** and choose **Point Load**.

2 Select Point 5 only.

Use only half the total load since you only model one symmetry half of the full geometry.

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

4 Specify the $\mathbf{F}_\mathrm{P}$ vector as

0	x
-Fn/2	y

Attach a spring to the cylinder in order to prevent rigid body motion before the contact is detected.

Spring Foundation 1

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

2 Select Point 5 only.

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

4 In the $\mathbf{k}_\mathrm{P}$ text field, type k.

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	E1	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu0	1	Young's modulus and Poisson's ratio
Density	rho	1	kg/m^3	Basic

Material 2 (mat2)

1. Right-click **Materials** and choose **Blank Material**.
2. Select Domain 2 only.
3. In the **Settings** window for **Material**, locate the **Material Contents** section.
4. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Young's modulus	E	E2	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	nu0	1	Young's modulus and Poisson's ratio
Density	rho	1	kg/m^3	Basic

The analytical solution to this problem assumes that engineering strains are used. Since the solution of a contact problem forces the study step to be geometrically nonlinear, you must explicitly enforce a linear strain representation.

SOLID MECHANICS (SOLID)

Linear Elastic Material 1

1. In the **Model Builder** window, under **Component 1 (comp1)>Solid Mechanics (solid)** click **Linear Elastic Material 1**.
2. In the **Settings** window for **Linear Elastic Material**, locate the **Geometric Nonlinearity** section.
3. From the **Formulation** list, choose **Geometrically linear**.

MESH 1

Free Triangular 1

1. In the **Mesh** toolbar, click **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Domain Selection** section.

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

4 Select Domain 2 only.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

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

4 Select Boundary 7 only.

5 Locate the **Element Size** section. Click the **Custom** button.

6 Locate the **Element Size Parameters** section.

7 Select the **Maximum element size** check box. In the associated text field, type `0.6`.

8 Click **Build All**.

Mapped 1

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

2 In the **Settings** window for **Mapped**, click to expand the **Control Entities** section.

3 Clear the **Smooth across removed control entities** check box.

Distribution 1

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

2 Select Boundaries 3, 10, and 11 only.

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

4 In the **Number of elements** text field, type `20`.

Distribution 2

1 In the **Model Builder** window, right-click **Mapped 1** and choose **Distribution**.

2 Select Boundary 1 only.

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

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

5 Click **Build All**.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** 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
k (Spring coefficient)	`Fn/dist/5 0`	`N/m`

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

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

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

RESULTS

Surface 1

1 In the **Model Builder** window, expand the **Results>Stress (solid)** node, then click **Surface 1**.

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

3 From the **Unit** list, choose **MPa**.

4 In the **Stress (solid)** toolbar, click **Plot**.

Because the point load gives a singular stress at the top of the cylinder, adjust the color range to see the stress distribution around the contact region better.

5 Click to expand the **Range** section. Select the **Manual color range** check box.

6 In the **Maximum** text field, type `2500`.

7 In the **Stress (solid)** toolbar, click **Plot**.

Contact Pressure

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

2 In the **Settings** window for **1D Plot Group**, type `Contact Pressure` in the **Label** text field.

3 Locate the **Data** section. From the **Parameter selection (k)** list, choose **Last**.

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

Line Graph 1

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

2 Select Boundary 7 only.

3 In the **Settings** window for **Line Graph**, click **Replace Expression** in the upper-right corner of the **y-Axis Data** section. From the menu, choose **Component 1 (comp1)>Definitions>Variables>p_analytical - Analytical contact pressure - N/m²**.

4 Locate the **y-Axis Data** section. From the **Unit** list, choose **MPa**.

5 Click **Replace Expression** in the upper-right corner of the **x-Axis Data** section. From the menu, choose **Component 1 (comp1)>Geometry>Coordinate (spatial frame)>x - x-coordinate**.

6 Click to expand the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Dashed**.

7 From the **Width** list, choose **2**.

8 Click to expand the **Legends** section. Select the **Show legends** check box.

9 From the **Legends** list, choose **Manual**.

10 In the table, enter the following settings:

Legends
`Analytical`

11 In the **Contact Pressure** toolbar, click **Plot**.

12 Right-click **Line Graph 1** and choose **Duplicate**.

Line Graph 2

1 In the **Model Builder** window, click **Line Graph 2**.

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

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

4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Line** list, choose **Solid**.

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

Legends
`Computed (Penalty)`

To avoid oscillations in the contact pressure representation, turn off the refinement within the elements.

6 Click to expand the **Quality** section. From the **Resolution** list, choose **No refinement**.

Contact Pressure

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

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

3 Select the **x-axis label** check box. In the associated text field, type `Distance from center (mm)`.

4 Select the **y-axis label** check box. In the associated text field, type `Contact pressure (MPa)`.

5 In the **Contact Pressure** toolbar, click **Plot**.

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

SOLID MECHANICS (SOLID)

Contact 1a

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 (p1)** 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 In the **Model Builder** window, right-click **Contact 1a** and choose **Duplicate**.

Contact 2

1 In the **Model Builder** window, click **Contact 2**.

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

Step 1: Stationary

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

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

3 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 2**.

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

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
k (Spring coefficient)	`Fn/dist/5 0`	`N/m`

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

9 In the **Settings** window for **Study**, locate the **Study Settings** section.

10 Clear the **Generate default plots** check box.

11 In the **Label** text field, type `Study 2: Augmented Lagrangian, Segregated`.

Solution 2 (sol2)

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

Adjust the scale for the contact pressure variable based on the analytical solution.

2 In the **Model Builder** window, expand the **Solution 2 (sol2)** node.

3 In the **Model Builder** window, expand the **Study 2: Augmented Lagrangian, Segregated>Solver Configurations>Solution 2 (sol2)>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 `1e9`.

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

The default plot for the second study was disabled. To visualize the stress and contact forces, change the dataset in the 2D plot group.

Similarly, add a third study for the augmented Lagrangian formulation with a coupled solution method and compute the solution.

ADD STUDY

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

STUDY 3

Step 1: Stationary

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
k (Spring coefficient)	`Fn/dist/5 0`	`N/m`

5 In the **Model Builder** window, click **Study 3**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 In the **Label** text field, type `Study 3: Augmented Lagrangian, Coupled`.

Solution 3 (sol3)

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

2 In the **Model Builder** window, expand the **Solution 3 (sol3)** node.

3 In the **Model Builder** window, expand the **Study 3: Augmented Lagrangian, Coupled>Solver Configurations>Solution 3 (sol3)>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 `1e9`.

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

RESULTS

Line Graph 2

In the **Model Builder** window, under **Results>Contact Pressure** right-click **Line Graph 2** and choose **Duplicate**.

Line Graph 3

1. In the **Model Builder** window, click **Line Graph 3**.
2. In the **Settings** window for **Line Graph**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 2: Augmented Lagrangian, Segregated/ Solution 2 (sol2)**.
4. From the **Parameter selection (k)** list, choose **Last**.
5. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Computed (Augmented Lagrangian, seg.)`

6. In the **Model Builder** window, right-click **Line Graph 3** and choose **Duplicate**.

Line Graph 4

1. In the **Model Builder** window, click **Line Graph 4**.
2. In the **Settings** window for **Line Graph**, locate the **Data** section.
3. From the **Dataset** list, choose **Study 3: Augmented Lagrangian, Coupled/Solution 3 (sol3)**.
4. From the **Parameter selection (k)** list, choose **Last**.
5. Locate the **Legends** section. In the table, enter the following settings:

Legends
`Computed (Augmented Lagrangian, cpl.)`

6. In the **Contact Pressure** toolbar, click **Plot**.

Now, solve the model using the Nitsche method.

SOLID MECHANICS (SOLID)

Contact 3

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 (p1)** in the **Pairs** list.

5 Click **OK**.

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

7 From the list, choose **Nitsche**.

Add a fourth study for the Nitsche method and compute the solution.

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

Step 1: Stationary

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

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

3 Click **Add**.

4 In the table, enter the following settings:

Parameter name	Parameter value list	Parameter unit
k (Spring coefficient)	`Fn/dist/5 0`	`N/m`

5 In the **Model Builder** window, click **Study 4**.

6 In the **Settings** window for **Study**, locate the **Study Settings** section.

7 Clear the **Generate default plots** check box.

8 In the **Label** text field, type `Study 4: Nitsche`.

Solution 4 (sol4)

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

2 Click **Compute**.

RESULTS

Line Graph 4

In the **Model Builder** window, under **Results>Contact Pressure** right-click **Line Graph 4** and choose **Duplicate**.

Line Graph 5

1 In the **Model Builder** window, click **Line Graph 5**.
2 In the **Settings** window for **Line Graph**, locate the **Data** section.
3 From the **Dataset** list, choose **Study 4: Nitsche/Solution 4 (sol4)**.
4 From the **Parameter selection (k)** list, choose **Last**.
5 Locate the **y-Axis Data** section. In the **Expression** text field, type `gpeval(8,solid.Tn)`.
6 Locate the **Legends** section. In the table, enter the following settings:

Legends
`Computed (Nitsche)`

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

Prepare the model for later use by making sure that the correct **Contact** node is active 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 1a**.
5 Right-click and choose **Disable**.
6 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 2**.
7 Right-click and choose **Disable**.
8 In the tree, select **Component 1 (comp1)>Solid Mechanics (solid), Controls spatial frame>Contact 3**.
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. 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**.

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 3**.
5. Right-click and choose **Disable**.