Self-Contact of a Loaded Spring

Introduction

The model in this example is an extension of *Loaded Spring — Using Global Equations to Satisfy Constraints* from the COMSOL Multiphysics Application Library. The same spring is here subjected to a load such that the spring contracts and self-intersects. The contact and friction introduce significant nonlinearity to the force-displacement response of the spring.

Model Definition

Figure 1 shows the geometry of the steel coil spring. The bottom end is fixed while a distributed load is applied to the other end, acting in the axial direction of the spring. As in the base model, the load is applied as a variable controlled by a global equation. The global equation here specifies that the displacement of the loaded end should be ramped linearly from 0 cm to –10 cm in the y direction.

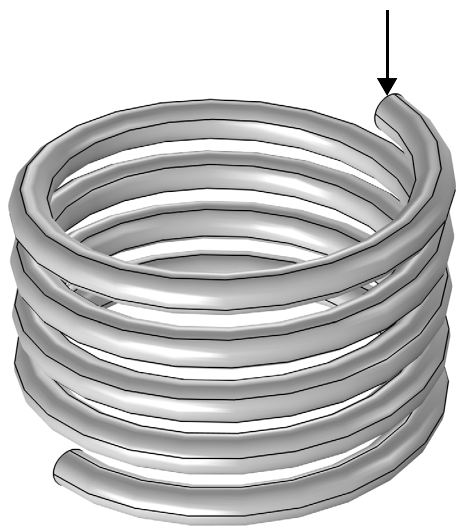

Figure 1: The geometry of the coil spring, also showing the application point of the load.

As the applied deformation is ramped, the contraction of the spring will reach a point where the spring will self-intersect. To prevent any unphysical overlap of the geometry, an additional contact condition specifies that all spring boundaries (except the end points) must not overlap. A friction condition is also added so that there is a resistance to relative motion in the tangential directions of any boundaries in contact. The Coulomb friction model is used with a friction coefficient equal to 0.45. Both the contact and friction conditions are here implemented by the Nitsche method.

Results and Discussion

Figure 2 shows the deformed shape of the spring at different vertical displacements. It can be clearly seen how the coil of the spring self-intersects and that the contact prevents any overlap. This, moreover, causes the spring to rotate around the contact points as the displacement is further increased. The vertical displacement at which the coil self-intersects is also clearly visible in Figure 3, which shows the force applied to the spring. At approximatively –4.5 cm, the is a clear change in the curve, which indicates that contact is established. After this point, it can be observed that the force versus displacement response becomes nonlinear, an effect of the slip between the contacting boundaries. If the displacement would be increased beyond 10 cm, the contact would eventually be lost as the friction resistance is overcome causing even greater nonlinearity to the response.

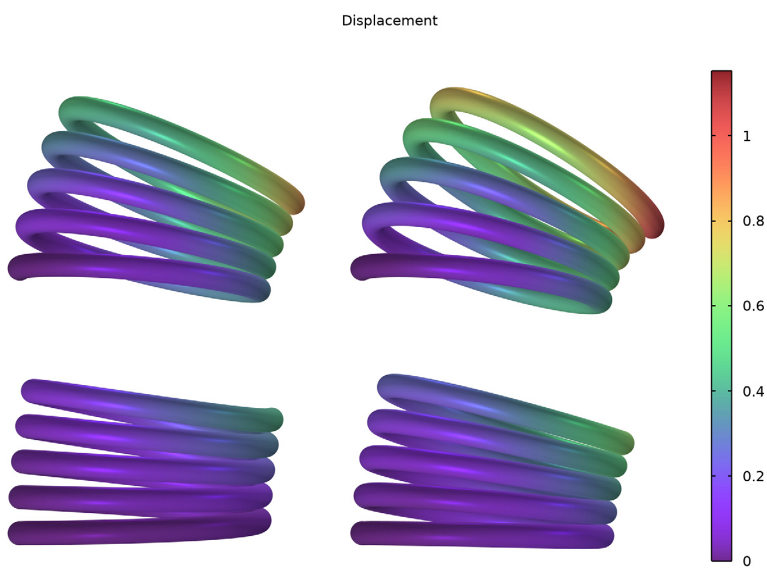

Figure 2: The deformed shape of the spring at a vertical displacement of -4 cm, -6 cm, -8 cm, and -10 cm.

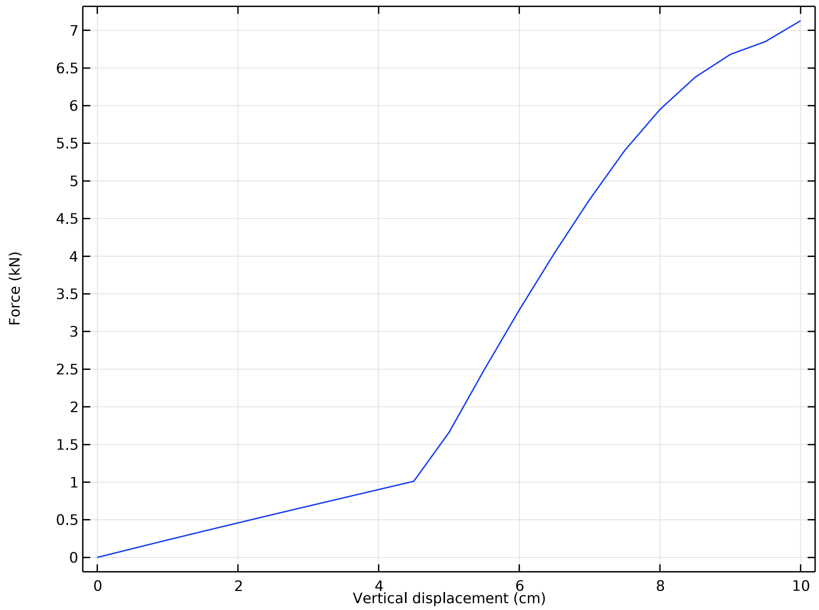

Figure 3: Force versus vertical displacement measured at the end point at the top of the spring.

Notes About the COMSOL Implementation

In COMSOL Multiphysics, self-contact is modeled by selecting the same boundaries as source and destination in a **Contact Pair**.

Application Library path: `Structural_Mechanics_Module/Contact_and_Friction/loaded_spring_contact`

Modeling Instructions

APPLICATION LIBRARIES

1. From the **File** menu, choose **Application Libraries**.
2. In the **Application Libraries** window, select **COMSOL Multiphysics>Structural Mechanics>loaded_spring** in the tree.

3 Click **Open**.

DEFINITIONS

Contact Pair 1 (p1)

1 In the **Model Builder** window, expand the **Component 1 (comp1)** node.

2 Right-click **Component 1 (comp1)>Definitions** and choose **Pairs>Contact Pair**.

To model self-contact, select the same boundaries as source and destination.

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

4 From the **Selection** list, choose **All boundaries**.

5 Select Boundaries 2–5 only.

6 Locate the **Destination Boundaries** section. From the **Selection** list, choose **All boundaries**.

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

8 Select Boundaries 2–5 only.

SOLID MECHANICS (SOLID)

Contact 1

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

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

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

The structural stiffness of spring is much lower than the bulk stiffness used to estimate the penalty factor. Using a lower penalty factor will improve the stability of the solution.

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 `0.01`.

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 `0.45`.

STUDY 1

Step 1: Stationary

1 In the **Model Builder** window, expand the **Study 1** node, then 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**.

In this example, the spring is compressed incrementally.

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
dh (Prescribed extension)	`-range(0,0.5,10)`	`cm`

Reset the solver to default since you are now solving a nonlinear problem.

Solution 1 (sol1)

In the **Model Builder** window, right-click **Solver Configurations** and choose **Show Default Solver**.

RESULTS

Stress (solid)

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

Deformation

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

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type `1`.

Stress (solid)

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

Plot the deformed spring at different solution parameters.

Displacement

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Displacement` in the **Label** text field.

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

4 Locate the **Plot Settings** section. Clear the **Plot dataset edges** check box.

5 Click to expand the **Plot Array** section. Select the **Enable** check box.

6 From the **Array shape** list, choose **Square**.

7 From the **Array plane** list, choose **xz**.

Volume 1

1 Right-click **Displacement** and choose **Volume**.

2 In the **Settings** window for **Volume**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (sol1)**.

4 From the **Parameter value (dh (cm))** list, choose **-4**.

5 Locate the **Coloring and Style** section. Click **Change Color Table**.

6 In the **Color Table** dialog box, select **Rainbow>SpectrumLight** in the tree.

7 Click **OK**.

Deformation 1

1 Right-click **Volume 1** and choose **Deformation**.

2 In the **Settings** window for **Deformation**, locate the **Scale** section.

3 Select the **Scale factor** check box. In the associated text field, type 1.

Volume 1

In the **Model Builder** window, right-click **Volume 1** and choose **Duplicate**.

Volume 2

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

2 In the **Settings** window for **Volume**, locate the **Data** section.

3 From the **Parameter value (dh (cm))** list, choose **-6**.

4 In the **Displacement** toolbar, click **Plot**.

5 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Volume 1**.

6 Right-click **Volume 2** and choose **Duplicate**.

Volume 3

1 In the **Model Builder** window, click **Volume 3**.

2 In the **Settings** window for **Volume**, locate the **Data** section.

3 From the **Parameter value (dh (cm))** list, choose **-8**.

4 Right-click **Volume 3** and choose **Duplicate**.

Volume 4

1 In the **Model Builder** window, click **Volume 4**.

2 In the **Settings** window for **Volume**, locate the **Data** section.

3 From the **Parameter value (dh (cm))** list, choose **-10**.

4 In the **Displacement** toolbar, click **Plot**.

Create a plot of the applied force versus the vertical displacement.

Force

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

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

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

Global 1

1 Right-click **Force** and choose **Global**.

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

3 In the table, enter the following settings:

Expression	Unit	Description
`-Force`	`kN`	`Force`

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

5 In the **Expression** text field, type `-dh`.

6 From the **Unit** list, choose **cm**.

7 Select the **Description** check box. In the associated text field, type `Vertical displacement`.

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

9 In the **Force** toolbar, click **Plot**.