Model created in COMSOL Multiphysics 6.4

# Contact Analysis of an Elastic Snap Hook

## *Introduction*

In this example, the insertion of a snap hook into a slot is modeled. The objective is to compute the force needed to place the hook in the slot. The problem thus involves modeling the contact between the hook and the lock during this process.

## *Model Definition*

The geometry of the model is shown in Figure 1. Due to the symmetry, you can study half of the original snap hook geometry.

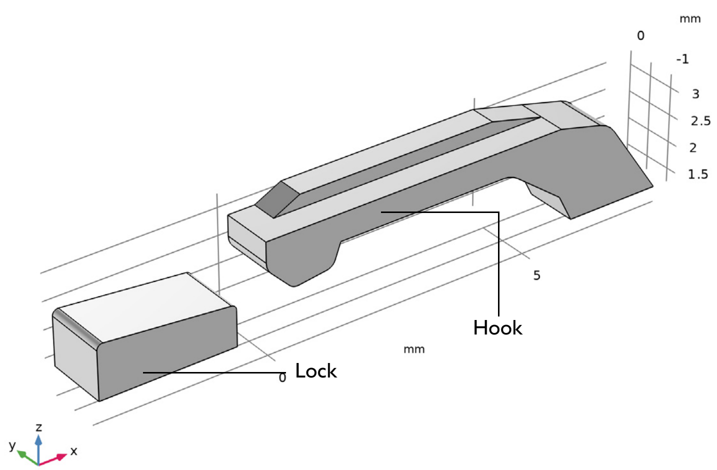


*Figure 1: Geometry of the modeled half of the snap hook and locking mechanism.*

#### MATERIAL PROPERTIES

The hook and lock are made of a modified nylon material. However, the lock is assumed rigid in this example. For the hook, a linear elastic material model is used, with material parameters given in the following table:

| MATERIAL PARAMETER | VALUE |
|---|---|
| Young's modulus | 10 GPa |
| Poisson's ratio | 0.35 |
| Density | 1150 kg/m$^3$ |

### BOUNDARY CONDITIONS

- The locking mechanism is considered as rigid, and is modeled as a meshed surface without physics.
- A prescribed displacement boundary condition is applied at the rightmost bottom surface of the hook. The displacement in the $x$ direction is gradually changed by using the parametric solver; the other two displacement components are zero.
- Two boundaries within the $xz$-plane use symmetry boundary conditions.
- All the other boundaries are free boundaries. However, several of them are selected as parts of a contact pair with the destination side being on the hook surface.

### CONTACT

- A contact pair is defined with boundaries on the lock selected as source, and boundaries on the hook selected as destination.
- Contact without friction is considered using the penalty method.
- The mesh on the source only needs to resolve the geometry of the contact surface. Hence no mesh is needed for the domain, and a refined mesh of the contact surface is only required for the rounded corners of the lock.

## *Results*

Figure 2 shows that the maximum penetration is less than 8 microns during the entire analysis. This is a good accuracy when compared to the geometry size.

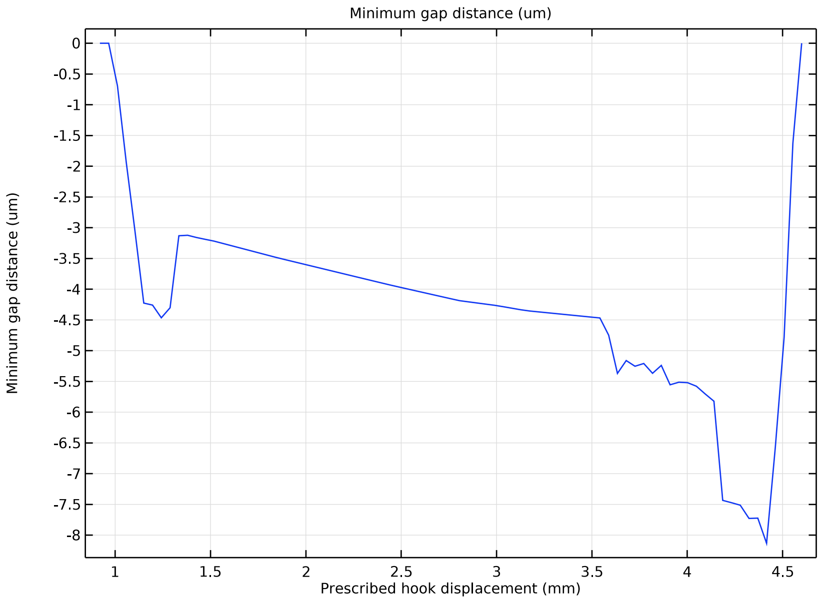


*Figure 2: Evolution of the maximum penetration for all position of the hook.*

Note that the penetration is not constant since the contact force varies depending on the position of the hook.

The maximum equivalent stress levels are found when the displacement of the hook is 3.8 mm, which is just before the hook enters the slot, see Figure 3.

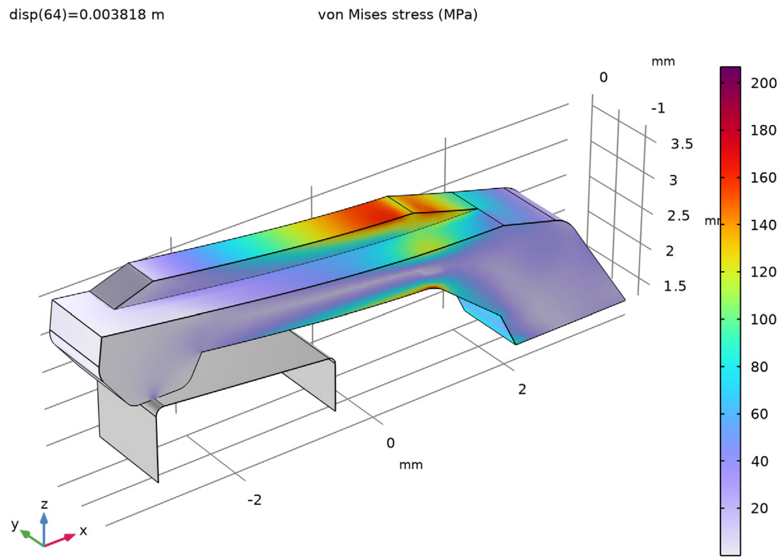


*Figure 3: The equivalent stress in the hook just before it enters the slot.*

[Figure 4](#) shows the force required for the insertion of the hook versus its prescribed displacement. The hook is in its slot at the end of the simulation.

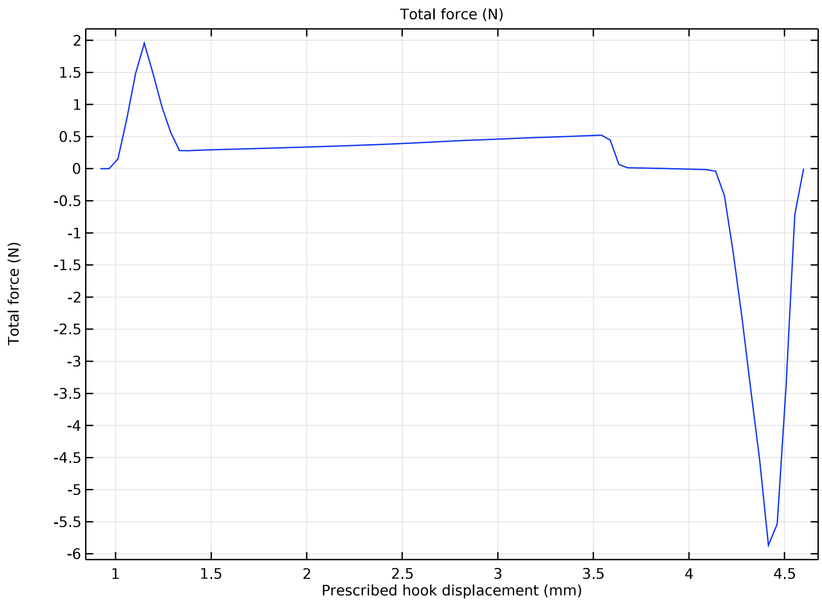


*Figure 4: The mounting force as a function of the hook displacement.*

---

**Application Library path:** `Structural_Mechanics_Module/Contact_and_Friction/snap_hook_elastic`

---

## *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 **3D**.

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 |
|---|---|---|---|
| `disp_max` | `4.6[mm]` | 0.0046 m | `Maximum hook displacement` |
| `disp` | `0[m]` | 0 m | `Prescribed hook displacement` |

## GEOMETRY 1

### *Import 1 (imp1)*

1 In the **Home** toolbar, click **Import**.

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

3 Click **Browse**.

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

5 Click **Import**.

### *Rotate 1 (rot1)*

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

2 Select the object **imp1** only.

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

4 From the **Axis type** list, choose **x-axis**.

5 In the **Angle** text field, type `90`.

6 Click **Build Selected**.

7 In the **Model Builder** window, click **Geometry 1**.

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

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

### *Scale 1 (sca1)*

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

2 Select the object **rot1** only.

3 In the **Settings** window for **Scale**, locate the **Scale Factor** section.

4 In the **Factor** text field, type `1000`.

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

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

## DEFINITIONS

### *Source Boundaries*

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Source Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select the **Group by continuous tangent** checkbox.

5 Select Boundaries 1, 4, and 6–8 only.

### *Destination Boundaries*

1 In the **Definitions** toolbar, click **Explicit**.

2 In the **Settings** window for **Explicit**, type `Destination Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Select Boundaries 14, 15, 20, 22, and 23 only.

### *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 From the **Selection** list, choose **Source Boundaries**.

4 Locate the **Destination Boundaries** section. Click to select the **Activate Selection** toggle button.

5 From the **Selection** list, choose **Destination Boundaries**.

## 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 | `10[GPa]` | Pa | Young's modulus and Poisson's ratio |
| Poisson's ratio | nu | `0.35` | 1 | Young's modulus and Poisson's ratio |
| Density | rho | `1150[kg/m^3]` | kg/m³ | Basic |

### 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 **Domain Selection** section.

**3** From the **Selection** list, choose **Manual**.

**4** Select Domains 2 and 3 only.

#### *Contact 1*

**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 Pressure Penalty Factor** section.

**3** From the **Penalty factor control** list, choose **Manual tuning**.

**4** In the $f_{\mathrm{p}}$ text field, type `1/10`.

#### *Prescribed Displacement 1*

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

**2** Select Boundary 30 only.

**3** In the **Settings** window for **Prescribed Displacement**, locate the **Prescribed Displacement** section.

**4** From the **Displacement in x direction** list, choose **Prescribed**.

**5** From the **Displacement in y direction** list, choose **Prescribed**.

**6** From the **Displacement in z direction** list, choose **Prescribed**.

**7** In the $u_{0x}$ text field, type `-disp`.

#### *Symmetry 1*

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

**2** Select Boundaries 13 and 19 only.

## MESH 1

Add a structured mesh on the contact destination boundaries.

### *Mapped 1*

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Destination Boundaries**.

### *Distribution 1*

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

2 Select Edges 27, 28, 39, and 48 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 Edges 31 and 45 only.

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

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

5 Click **Build Selected**.

### *Convert 1*

Convert the quad mesh to triangles so that the rest of the geometry can be meshed using the free tetrahedral method.

1 In the **Mesh** toolbar, click **Modify** and choose **Convert**.

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

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

4 From the **Selection** list, choose **Destination Boundaries**.

5 Click **Build Selected**.

### *Free Tetrahedral 1*

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

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

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

4 Select Domains 2 and 3 only.

*Size 1*

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

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** checkbox. In the associated text field, type `0.2`.

*Size 2*

1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.

Refine the mesh on the boundary where high stresses are expected.

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 25 only.

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

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

7 Select the **Maximum element size** checkbox. In the associated text field, type `0.1`.

8 Click **Build Selected**.

*Mapped 2*

Add a surface mesh for the lock. Notice that no mesh is needed for the domain.

1 In the **Mesh** toolbar, click **More Generators** and choose **Mapped**.

2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **Source Boundaries**.

*Size 1*

1 Right-click **Mapped 2** and choose **Size**.

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

3 From the **Predefined** list, choose **Extremely coarse**.

*Distribution 1*

1 In the **Model Builder** window, right-click **Mapped 2** and choose **Distribution**.

2 Select Edges 5 and 13 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*

Set up an auxiliary continuation sweep for the `disp` 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 **Study Extensions** section.

3 Select the **Auxiliary sweep** checkbox.

4 Click **Add**.

5 In the table, enter the following settings:

| Parameter name | Parameter value list | Parameter unit |
|---|---|---|
| disp (Prescribed hook displacement) | `range(0.2,1e-2,1)*disp_max` | m |

6 Click to expand the **Results While Solving** section. Select the **Plot** checkbox.

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

## RESULTS

*Stress (solid)*

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Parameter value (disp (m))** list, choose **0.003818**.

*Volume 1*

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

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

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

*Stress (solid)*

Add a surface plot for the lock. You can write an arbitrary value in the expression field since a uniform color is used.

*Surface 1*

1 In the **Model Builder** window, right-click **Stress (solid)** and choose **Surface**.

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

3 In the **Expression** text field, type `1`.

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

5 Click to collapse the **Title** section. Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Gray**.

### *Selection 1*

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Source Boundaries**.

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

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

## RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Solid Mechanics** > **Contact Forces (solid)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

## RESULTS

### *Contact Forces (solid)*

1 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

2 From the **Parameter value (disp (m))** list, choose **0.003818**.

### *Minimum Gap Distance*

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

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

3 Locate the **Legend** section. Clear the **Show legends** checkbox.

### *Global 1*

1 In the **Minimum Gap Distance** toolbar, click **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 |
|---|---|---|
| solid.gapmin_p1*(solid.gapmin_p1<0) | um | Minimum gap distance |

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

5 In the **Expression** text field, type `disp`.

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

7 In the **Minimum Gap Distance** toolbar, click **Plot**.

### *Reaction Force*

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

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

3 Locate the **Legend** section. Clear the **Show legends** checkbox.

### *Global 1*

1 In the **Reaction Force** toolbar, click **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 |
|---|---|---|
| -2*solid.disp1.RFsumx | N | Total force |

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

5 In the **Expression** text field, type `disp`.

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

7 In the **Reaction Force** toolbar, click **Plot**.