

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 a half of the original snap hook geometry.

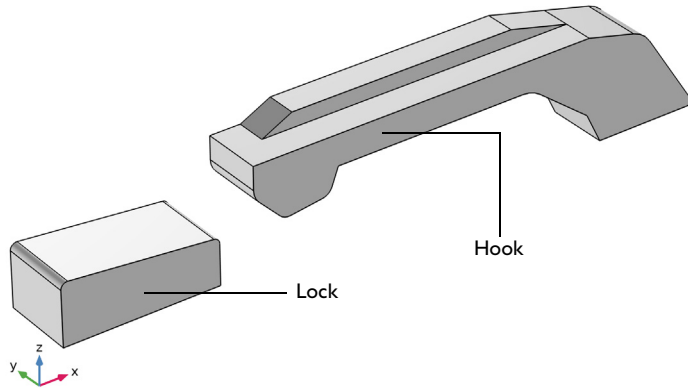


Figure 1: Geometry of the modeled half of the snap hook and locking mechanism.

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 side 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.

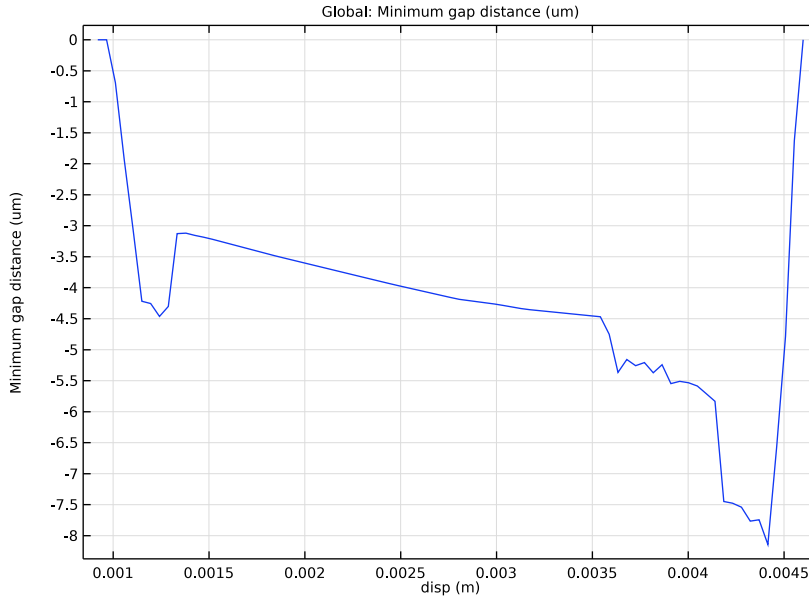


Figure 2: Evolution of the maximum penetration for all position of the hook.

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](#).

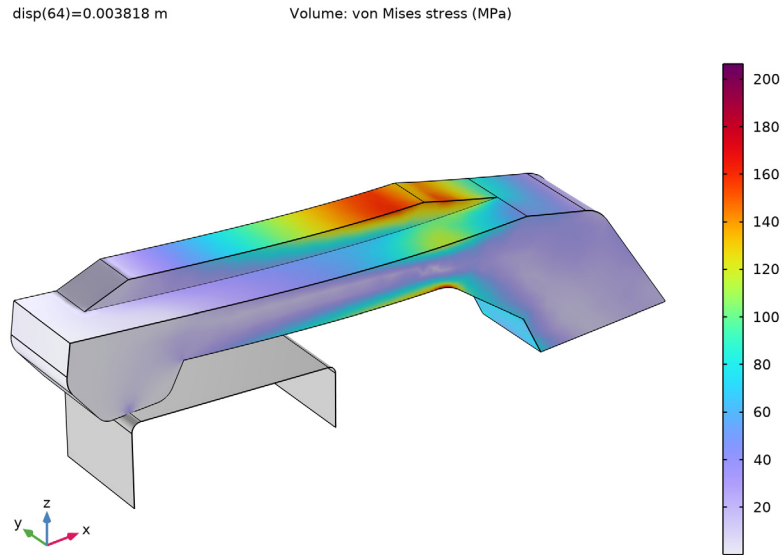


Figure 3: The equivalent stress levels in the hook just before it enters the slot.

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.

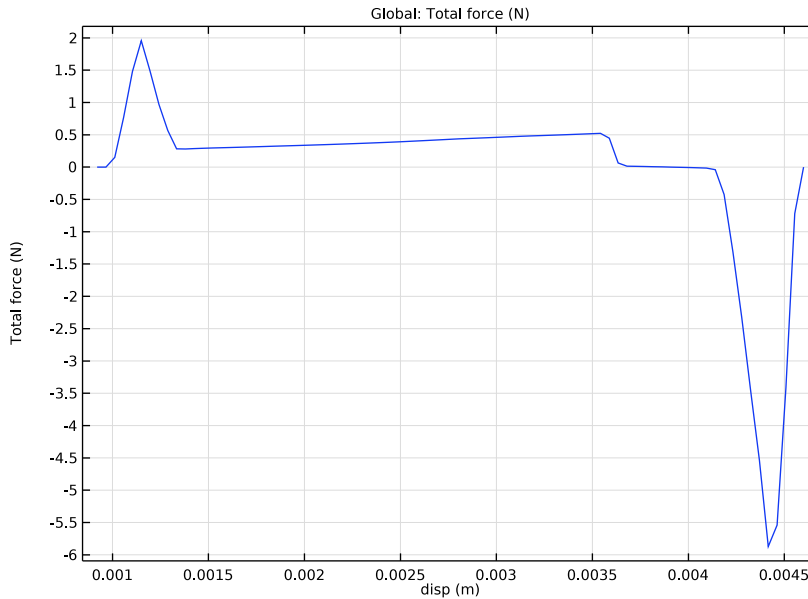


Figure 4: The mounting force as a function of the hook displacement.

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 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
Displ_max	4.6[mm]	0.0046 m	Maximum hook displacement
disp	0[m]	0 m	Prescribed hook displacement

GEOMETRY I

Import I (impI)

- 1 In the **Home** toolbar, click  **Import**.
- 2 In the **Settings** window for **Import**, locate the **Import** 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 I (rotI)

- 1 In the **Geometry** toolbar, click  **Transforms** and choose **Rotate**.
- 2 Select the object **impI** only.
- 3 In the **Settings** window for **Rotate**, locate the **Rotation** section.
- 4 In the **Angle** text field, type 90.
- 5 From the **Axis type** list, choose **Cartesian**.
- 6 In the **x** text field, type 1.
- 7 In the **z** text field, type 0.
- 8 Click  **Build Selected**.
- 9 Click the  **Zoom Extents** button in the **Graphics** toolbar.

DEFINITIONS

contact_src

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type *contact_src* 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** check box.
- 5 Select Boundaries 1, 4, and 6–8 only.

contact_dst

- 1 In the **Definitions** toolbar, click  **Explicit**.
- 2 In the **Settings** window for **Explicit**, type *contact_dst* 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 (pl)

- 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 *contact_src*.
- 4 Locate the **Destination Boundaries** section. Click to select the  **Activate Selection** toggle button.
- 5 From the **Selection** list, choose *contact_dst*.

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_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 Select the **Prescribed in x direction** check box.
- 5 Select the **Prescribed in y direction** check box.
- 6 Select the **Prescribed in z direction** check box.
- 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  **Boundary** and choose **Mapped**.
- 2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **contact_dst**.

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 **contact_dst**.
- 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** check box. In the associated text field, type 4e-4.

Size 2

- 1 In the **Model Builder** window, right-click **Free Tetrahedral 1** and choose **Size**.
Refine the mesh on 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** check box. In the associated text field, type 1e-4.
- 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  **Boundary** and choose **Mapped**.
- 2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.
- 3 From the **Selection** list, choose **contact_src**.

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**.

Before setting the study, add a nonlocal integration coupling that will be used for postprocessing the reaction force.

DEFINITIONS

Integration 1 (intop1)

- 1 In the **Definitions** toolbar, click  **Nonlocal Couplings** and choose **Integration**.
- 2 In the **Settings** window for **Integration**, locate the **Source Selection** section.
- 3 From the **Geometric entity level** list, choose **Boundary**.
- 4 Select Boundary 30 only.

5 Locate the **Advanced** section. From the **Method** list, choose **Summation over nodes**.

STUDY I

Step 1: Stationary

Set up an auxiliary continuation sweep for the **disp** parameter.

- 1 In the **Model Builder** window, under **Study I** 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
disp (Prescribed hook displacement)	range(0.2, 1e-2, 1) * Displ_max	m

- 6 Click to expand the **Results While Solving** section. Select the **Plot** check box.
- 7 In the **Home** 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**.
- 3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

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 I

- 1 Right-click **Surface I** and choose **Selection**.
- 2 In the **Settings** window for **Selection**, locate the **Selection** section.
- 3 From the **Selection** list, choose **contact_src**.
- 4 In the **Stress (solid)** toolbar, click  **Plot**.
- 5 Click the  **Zoom Extents** button in the **Graphics** toolbar.
- 6 In the **Home** toolbar, click  **Add Predefined Plot**.

ADD PREDEFINED PLOT

- 1 Go to the **Add Predefined Plot** window.
- 2 In the tree, select **Study I/Solution I (sol1)>Solid Mechanics>Contact Forces (solid)**.
- 3 Click **Add Plot** in the window toolbar.
- 4 In the **Home** toolbar, click  **Add Predefined Plot**.

RESULTS

Contact Forces (solid)

- 1 In the **Model Builder** window, under **Results** click **Contact Forces (solid)**.
- 2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.
- 3 From the **Parameter value (disp (m))** list, choose **0.003818**.

Minimum Gap Distance

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Minimum Gap Distance in the **Label** text field.
- 3 Locate the **Legend** section. Clear the **Show legends** check box.

Global I

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

Reaction Force

- 1 In the **Home** toolbar, click  **Add Plot Group** and choose **ID Plot Group**.
- 2 In the **Settings** window for **ID Plot Group**, type Reaction Force in the **Label** text field.
- 3 Locate the **Legend** section. Clear the **Show legends** check box.

Global I

- 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*intop1(solid.RFx)	N	Total force

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