

Contact Analysis of an Elastic Snap Hook

Introduction

In this example, the insertion of a snap hook in to 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 hook 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.
- Since no physics is defined on the look, it is in the **Contact** node considered as external to the current physics.
- 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: 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

I In the Model Wizard window, click 间 3D.

2 In the Select Physics tree, select Structural Mechanics>Solid Mechanics (solid).

- 3 Click Add.
- 4 Click \bigcirc Study.
- 5 In the Select Study tree, select General Studies>Stationary.
- 6 Click 🗹 Done.

GLOBAL DEFINITIONS

Parameters 1

- I 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	O[m]	0 m	Prescribed hook displacement

GEOMETRY I

Import I (imp1)

- I 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)

- I In the Geometry toolbar, click 💭 Transforms and choose Rotate.
- 2 Select the object impl 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 틤 Build Selected.
- **9** Click the **Com Extents** button in the **Graphics** toolbar.

DEFINITIONS

contact_src

- I 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

- I 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 I (p1)

- I In the **Definitions** toolbar, click **H 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. Select the **Destination** toggle button.
- 5 From the Selection list, choose contact_dst.

MATERIALS

Material I (mat1)

- I In the Model Builder window, under Component I (compl) 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	Basic
Poisson's ratio	nu	0.35	1	Basic
Density	rho	1150[kg/m^3]	kg/m³	Basic

SOLID MECHANICS (SOLID)

- I In the Model Builder window, under Component I (compl) 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.

Prescribed Displacement 1

- I 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 I

- I In the Physics toolbar, click 🔚 Boundaries and choose Symmetry.
- **2** Select Boundaries 13 and 19 only.

Contact I

- I In the Physics toolbar, click 🔚 Pairs and choose Contact.
- 2 In the Settings window for Contact, locate the Contact Surface section.
- **3** Select the **Source external to current physics** check box.
- 4 Locate the Pair Selection section. Under Pairs, click + Add.
- 5 In the Add dialog box, select Contact Pair I (pl) in the Pairs list.
- 6 Click OK.
- 7 In the Settings window for Contact, locate the Contact Pressure Penalty Factor section.
- 8 From the Penalty factor control list, choose Manual tuning.
- **9** In the f_p text field, type 1/10.

MESH I

Add a structured mesh on the contact destination boundaries.

Mapped I

I In the Mesh toolbar, click \bigwedge 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 I

- I Right-click Mapped I 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

- I In the Model Builder window, right-click Mapped I 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 I

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

- I In the Mesh toolbar, click A Modify and choose Elements>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 I

- I In the Mesh toolbar, click \land 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 I

- I Right-click Free Tetrahedral I 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. Select the Maximum element size check box.

5 In the associated text field, type 4e-4.

Size 2

- I In the **Model Builder** window, right-click **Free Tetrahedral I** 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. Select the Maximum element size check box.
- 7 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.

- I In the Mesh toolbar, click \bigwedge 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 I

- I 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 I

- I 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)

- I 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.

- I In the Model Builder window, under Study I click Step I: 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	range(0.2,1e-2,1)*	m
displacement)	Displ_max	

- 6 Click to expand the **Results While Solving** section. Select the **Plot** check box.
- **7** In the **Home** toolbar, click **= Compute**.

RESULTS

Stress (solid)

- I 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).

Surface 1

- I In the Model Builder window, expand the Stress (solid) node, then click Surface I.
- 2 In the Settings window for Surface, locate the Expression section.
- 3 From the Unit list, choose MPa.

Stress (solid)

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

Surface 2

- I 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

- I Right-click Surface 2 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 **I** Plot.
- **5** Click the \leftrightarrow **Zoom Extents** button in the **Graphics** toolbar.

Contact Forces (solid)

- I In the Model Builder window, 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

- I 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

- I 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 **O** Plot.

Reaction Force

- I 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

- I 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)	Ν	Total Force

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