Model created in COMSOL Multiphysics 6.4

Contact Switch

Introduction

A contact switch is used to regulate whether or not an electric current is passing from a power source into an electrical device. These switches are found in many types of equipment and they are used to control, for example, the power output from a wall socket into a device when it is plugged in; the currents passing across the circuit board of a computer; or the electricity powering a light bulb when the switch is flipped on. Because of their prevalence, simulating contact switches is a fundamental step in designing electronic applications.

The working principle behind a contact switch is simple: two conductive pieces of metal with an electric-voltage difference across them are brought into contact, allowing a current to flow between them. The metallic surfaces of the two components that touch one another are called contacts, and when the connection between the two contacts is broken, the current stops flowing.

The current flow between the two contacts contributes to an increase in temperature in the switch due to the Joule heating effect.

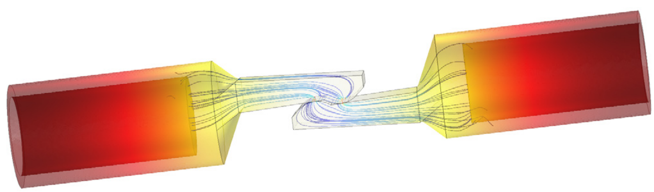

Figure 1: A contact switch.

The heating of the contact switch can change the material properties of the metal as well as the surface area of contact, and therefore is an important effect to consider when modeling the switch. Letting the temperature become too high can even cause the switch to burn out, meaning the switch is no longer functional. Therefore, it is important to analyze its current-carrying capability in order to prevent overheating. It is also important to consider that when the two metallic pieces come into contact, the surfaces touching each other experience a mechanical pressure or contact pressure. This mechanical pressure on the contacts can alter the electrical and thermal properties of the material locally around the region surrounding the contacts. Therefore, in order to accurately simulate the current-carrying capability and temperature rise in the switch, it is important to take a more comprehensive approach in the simulation and incorporate the effect of contact pressure to compute the electrical and thermal conductance of the contact surfaces.This tutorial illustrates how to implement a multiphysics contact. It models the thermal and electrical behavior of two contacting parts of a switch. The electric current and the heat cross from one part to the other only through the contact area.

The contact switch device has a cylindrical body and plate hook shapes at the contact area (see Figure 1). There, the thermal and electrical apparent resistances are coupled to the mechanical contact pressure at the interface, which the application solves.

The initial temperature is equal to the external room temperature. A potential difference between the left and right parts leads to heating through the Joule effect.

Model Definition

The geometry of the switch is shown in Figure 2. Only half of the device is represented due to symmetry considerations.

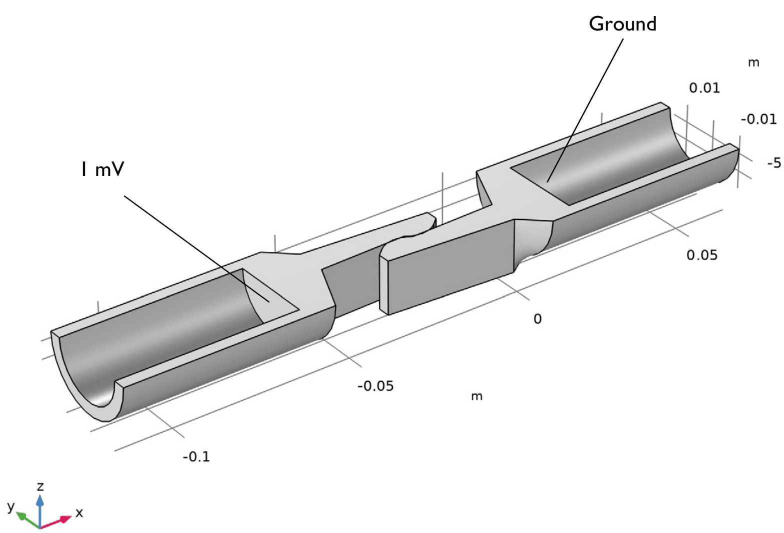

Figure 2: Switch geometry.

The switch is made of copper, with two fixed cylindrical elements and a central region where the contacts are located. On the end of each contact are plate hooks that enable contact between the two pieces. In the simulation, an electric potential of 1 mV is applied to the left side of the switch, while the right side is grounded.

The thermal and electrical contact conductances are assumed to be related only to the contact pressure.

The exposed surfaces of the switch lose heat due to their interaction with air via natural convection. In the simulation, this is modeled by specifying a heat transfer coefficient and the ambient temperature of the surrounding air (a more ambitious simulation might also include the fluid flow of the air). The application first solves for structural contact to obtain the contact pressure on the contact surfaces. These results are then used to compute the electrical and thermal conductance of the contact's surfaces in a Joule heating simulation.

Results and Discussion

Figure 3 shows the electric potential distribution, ranging from the grounded right side to the applied 1 mV on the left.

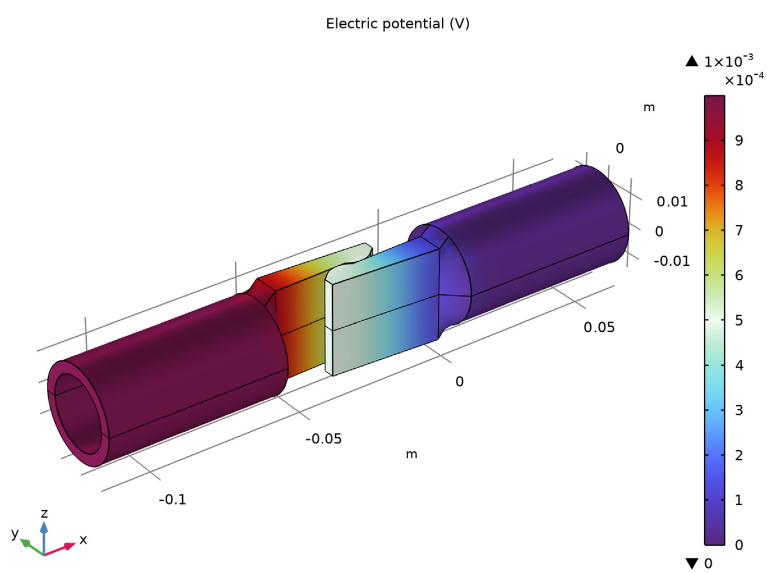

Figure 3: Electric potential profile.

A potential difference across the two components in the switch creates a current flow, which in turn leads to Joule heating. This causes a rise in temperature in the switch. If you leave the switch on for a while, temperature distribution in the switch reaches an equilibrium. Figure 4 shows the temperature distribution in the contact switch. In this example, Joule heating causes the temperature in the switch to rise about 5 K above room temperature, although only a small temperature variation is seen within the switch itself.

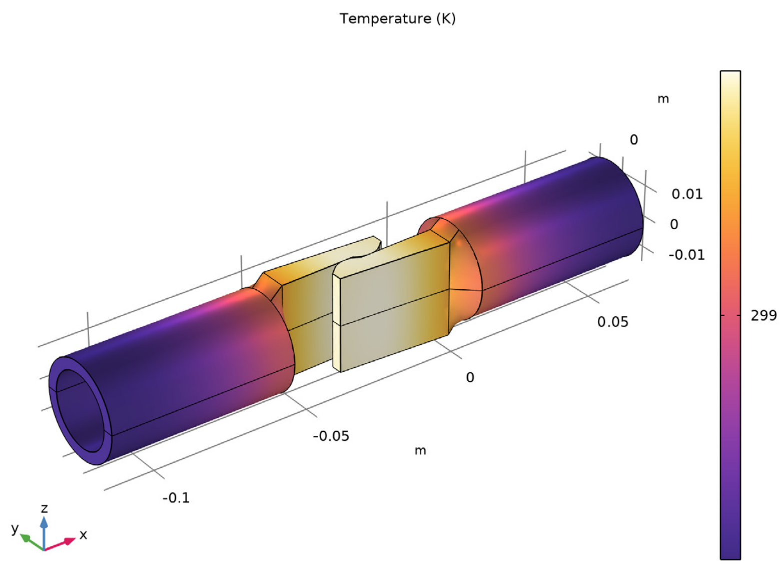

Figure 4: Temperature distribution.

The internal temperature distribution is almost constant. Introducing the effect of electrical and thermal conductance allows us to predict the temperature rise more accurately. The simulation also shows that the switch gets slightly hotter at the contact region.

Finally, Figure 5 plots the temperature distribution at the contact region. Streamlines show the current density.

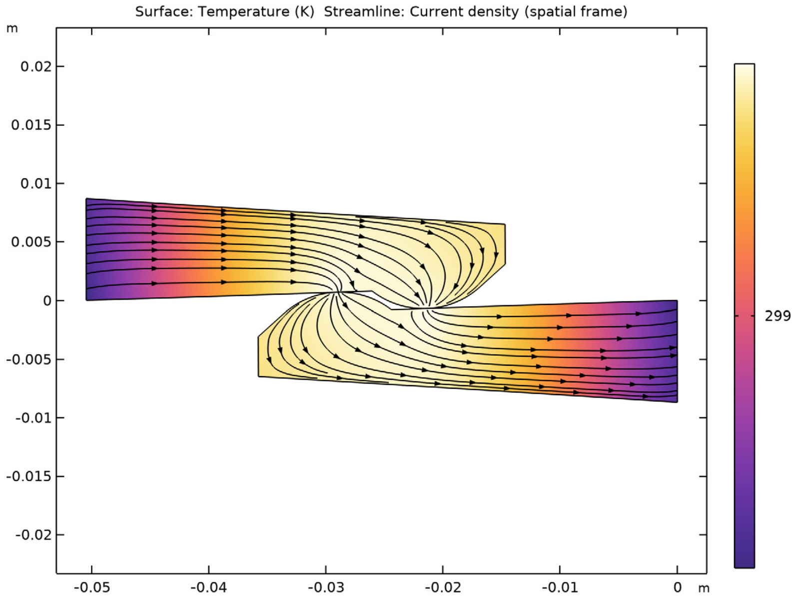

Figure 5: Temperature distribution (surface plot) and current density (streamlines) at the contact region.

Application Library path: `Heat_Transfer_Module/Thermal_Contact_and_Friction/contact_switch`

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 In the **Select Physics** tree, select **Heat Transfer** > **Electromagnetic Heating** > **Joule Heating**.

5 Click **Add**.

6 Click **Study**.

7 In the **Select Study** tree, select **General Studies** > **Stationary**.

8 Click **Done**.

GEOMETRY 1

The model geometry is provided in a separate MPHBIN file. If you prefer to create it from scratch, follow the steps outlined in the Geometry Modeling Instructions. Note that a license for the Design Module is required to create the geometry. Otherwise, you can import the geometry as follows:

Import 1 (imp1)

1 In the **Geometry** 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 `contact_switch.mphbin`.

5 Click **Import**.

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 From the **Pair type** list, choose **Contact pair**.

5 Clear the **Create pairs** checkbox.

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

DEFINITIONS

Contact Pair 1 (p1)

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

2 Select Boundaries 12 and 15 only.

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

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

5 Select Boundaries 25 and 28 only.

ADD MATERIAL

1 In the **Materials** toolbar, click **Add Material** to open the **Add Material** window.

2 Go to the **Add Material** window.

3 In the tree, select **Built-in** > **Copper**.

4 Click the **Add to Component** button in the window toolbar.

5 In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

SOLID MECHANICS (SOLID)

Fixed Constraint 1

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

2 Select Boundaries 4, 5, 34, and 35 only.

Symmetry 1

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

2 Select Boundaries 2 and 22 only.

Contact 1

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

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

3 From the list, choose **Augmented Lagrangian**.

HEAT TRANSFER IN SOLIDS (HT)

In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids (ht)**.

Heat Flux 1

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

2 Select Boundaries 3 and 6–33 only.

3 In the **Settings** window for **Heat Flux**, locate the **Heat Flux** section.

4 From the **Flux type** list, choose **Convective heat flux**.

5 In the h text field, type `2[W/(m^2*K)]`.

Symmetry 1

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

2 Select Boundaries 2 and 22 only.

Thermal Contact 1

1. In the **Physics** toolbar, click **Pairs** and choose **Thermal Contact**.
2. In the **Settings** window for **Thermal Contact**, locate the **Pair Selection** section.
3. Click **Add**.
4. In the **Add** dialog, select **Contact Pair 1 (p1)** in the **Pairs** list.
5. Click **OK**.
6. In the **Settings** window for **Thermal Contact**, locate the **Contact Surface Properties** section.
7. From the p list, choose **Contact pressure (solid/dcnt1)**.

ELECTRIC CURRENTS (EC)

In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents (ec)**.

Ground 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Ground**.
2. Select Boundary 34 only.

Electric Potential 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Electric Potential**.
2. Select Boundary 5 only.
3. In the **Settings** window for **Electric Potential**, locate the **Electric Potential** section.
4. In the V_0 text field, type `1[mV]`.

Electrical Contact 1

1. In the **Physics** toolbar, click **Pairs** and choose **Electrical Contact**.
2. In the **Settings** window for **Electrical Contact**, locate the **Pair Selection** section.
3. Click **Add**.
4. In the **Add** dialog, select **Contact Pair 1 (p1)** in the **Pairs** list.
5. Click **OK**.
6. In the **Settings** window for **Electrical Contact**, locate the **Contact Surface Properties** section.
7. From the p list, choose **Contact pressure (solid/dcnt1)**.

MESH 1

Free Tetrahedral 1

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

Size 1

1 Right-click **Free Tetrahedral 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 Boundaries 12, 15, 25, and 28 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 `5e-4`.

Size

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Mesh 1** click **Size**.

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

3 From the **Predefined** list, choose **Fine**.

4 Click **Build All**.

STUDY 1

Solve the model in two steps. The first step only computes for **Solid Mechanics** while the second solves for Joule Heating (**Electric Currents** and **Heat Transfer in Solids**).

Step 1: Stationary

1 In the **Model Builder** window, under **Study 1** click **Step 1: Stationary**.

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

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkboxes for **Electric Currents (ec)** and **Heat Transfer in Solids (ht)**.

4 In the **Solve for** column of the table, under **Component 1 (comp1)** > **Multiphysics**, clear the checkbox for **Electromagnetic Heating 1 (emh1)**.

Step 2: Stationary 2

1 In the **Study** toolbar, click **Stationary**.

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

3 In the **Solve for** column of the table, under **Component 1 (comp1)**, clear the checkbox for **Solid Mechanics (solid)**.

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

RESULTS

Stress (solid)

In this first default plot, the switch is slightly deformed due to the contact pressure. The von Mises stress is located at the switch base and at the contact area.

Follow the next steps to visualize the third and fifth default plots as in Figure 3 and Figure 4.

Mirror 3D 1

1 In the **Model Builder** window, expand the **Results** > **Datasets** node.

2 Right-click **Results** > **Datasets** and choose **More 3D Datasets** > **Mirror 3D**.

3 In the **Settings** window for **Mirror 3D**, locate the **Plane Data** section.

4 From the **Plane** list, choose **xy-planes**.

Electric Potential (ec)

1 In the **Model Builder** window, under **Results** click **Electric Potential (ec)**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 In the **Electric Potential (ec)** toolbar, click **Plot**.

Temperature (ht)

1 In the **Model Builder** window, click **Temperature (ht)**.

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

3 From the **Dataset** list, choose **Mirror 3D 1**.

4 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

5 In the **Temperature (ht)** toolbar, click **Plot**.

To observe the temperature and current density only at the contact region (Figure 5), proceed as follows.

Surface 1

1 In the **Results** toolbar, click **More Datasets** and choose **Surface**.

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

3 From the **x- and y-axes** list, choose **xy-plane**.

4 Select Boundaries 10 and 21 only.

Temperature (Contact Region)

1 In the **Results** toolbar, click **2D Plot Group**.

2 In the **Settings** window for **2D Plot Group**, type `Temperature (Contact Region)` in the **Label** text field.

3 Locate the **Plot Settings** section. From the **Frame** list, choose **Spatial (x, y, z)**.

Surface 1

1 In the **Temperature (Contact Region)** toolbar, click **Surface**.

2 In the **Settings** window for **Surface**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Heat Transfer in Solids** > **Temperature** > **T - Temperature - K**.

3 Locate the **Coloring and Style** section. From the **Color table** list, choose **HeatCameraLight**.

Temperature (Contact Region)

In the **Model Builder** window, click **Temperature (Contact Region)**.

Streamline 1

1 In the **Temperature (Contact Region)** toolbar, click **Streamline**.

2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1 (comp1)** > **Electric Currents** > **Currents and charge** > **ec.Jx,ec.Jy - Current density (spatial frame)**.

3 Locate the **Streamline Positioning** section. From the **Positioning** list, choose **Uniform density**.

4 In the **Density level** text field, type `8`.

5 Locate the **Coloring and Style** section. Find the **Point style** subsection. From the **Type** list, choose **Arrow**.

6 In the **Temperature (Contact Region)** toolbar, click **Plot**.

Geometry Modeling Instructions

Follow these steps to create the geometry for the contact switch model. Note that a license for the Design Module is required for some geometric operations.

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 Click **Done**.

GEOMETRY 1

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

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

3 From the **Geometry representation** list, choose **CAD kernel**.

4 Select the **Design Module Boolean operations** checkbox.

Work Plane 1 (wp1)

In the **Geometry** toolbar, click **Work Plane**.

Work Plane 1 (wp1) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 1 (wp1) > Polygon 1 (pol1)

1 In the **Work Plane** toolbar, click **Polygon**.

2 In the **Settings** window for **Polygon**, locate the **Coordinates** section.

3 In the table, enter the following settings:

xw (m)	**yw (m)**
-24.4[mm]	-0.68[mm]
0[mm]	0[mm]
0[mm]	-8.7[mm]
-35.8[mm]	-6.3[mm]
-35.8[mm]	-2.9[mm]
-33.6[mm]	-0.94[mm]

4 Locate the **Object Type** section. From the **Type** list, choose **Open curve**.

5 Click **Build Selected**.

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

Work Plane 1 (wp1) > Circular Arc 1 (ca1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Properties** section.

3 From the **Specify** list, choose **Endpoints and radius**.

4 Locate the **Starting Point** section. In the **xw** text field, type `-33.6[mm]`.

5 In the **yw** text field, type `-0.94[mm]`.

6 Locate the **Endpoint** section. In the **xw** text field, type `-24.4[mm]`.

7 In the **yw** text field, type `-0.68[mm]`.

8 Locate the **Radius** section. In the **Radius** text field, type `7.2[mm]`.

9 Locate the **Angles** section. Select the **Clockwise** checkbox.

10 Click **Build Selected**.

Work Plane 1 (wp1) > Convert to Solid 1 (csol1)

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

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

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

4 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

Work Plane 1 (wp1)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Work Plane 1 (wp1)** node.

Extrude 1 (ext1)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
`14.85[mm]`

4 Select the **Reverse direction** checkbox.

5 Click **Build Selected**.

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

Work Plane 2 (wp2)

1 In the **Geometry** toolbar, click **Work Plane**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 From the **Plane** list, choose **yz-plane**.

4 In the **x-coordinate** text field, type `6.8[mm]`.

Work Plane 2 (wp2) > Plane Geometry

In the **Model Builder** window, click **Plane Geometry**.

Work Plane 2 (wp2) > Circular Arc 1 (ca1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Circular Arc**.

2 In the **Settings** window for **Circular Arc**, locate the **Radius** section.

3 In the **Radius** text field, type `15[mm]`.

4 Locate the **Angles** section. In the **Start angle** text field, type `-180[deg]`.

5 In the **End angle** text field, type `0[deg]`.

6 Click **Build Selected**.

Work Plane 2 (wp2) > Line Segment 1 (ls1)

1 In the **Work Plane** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **ca1**, select Point 1 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 Click to select the **Activate Selection** toggle button for **End vertex**.

5 On the object **ca1**, select Point 2 only.

6 Click **Build Selected**.

Work Plane 2 (wp2) > Convert to Solid 1 (csol1)

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

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

3 In the **Settings** window for **Convert to Solid**, click **Build Selected**.

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

Work Plane 2 (wp2)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Work Plane 2 (wp2)** node.

Extrude 2 (ext2)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
58.5[mm]

4 Click **Build Selected**.

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

Work Plane 3 (wp3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 2 (wp2)** and choose **Duplicate**.

2 In the **Settings** window for **Work Plane**, locate the **Plane Definition** section.

3 In the **x-coordinate** text field, type `14.5[mm]`.

Work Plane 3 (wp3) > Plane Geometry

In the **Model Builder** window, expand the **Work Plane 3 (wp3)** node, then click **Plane Geometry**.

Work Plane 3 (wp3) > Circular Arc 1 (ca1)

1 In the **Model Builder** window, expand the **Work Plane 3 (wp3)** > **Plane Geometry** node, then click **Circular Arc 1 (ca1)**.

2 In the **Settings** window for **Circular Arc**, locate the **Radius** section.

3 In the **Radius** text field, type `11.45[mm]`.

Work Plane 3 (wp3)

In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Work Plane 3 (wp3)** node.

Extrude 3 (ext3)

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Extrude**.

2 In the **Settings** window for **Extrude**, locate the **Distances** section.

3 In the table, enter the following settings:

Distances (m)
50.8[mm]

4 Click **Build Selected**.

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

Difference 1 (dif1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Difference**.

2 Select the object **ext2** only.

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

4 Click to select the **Activate Selection** toggle button for **Objects to subtract**.

5 Select the object **ext3** only.

6 Click **Build Selected**.

Line Segment 1 (ls1)

1 In the **Geometry** toolbar, click **More Primitives** and choose **Line Segment**.

2 On the object **ext1**, select Point 11 only.

3 In the **Settings** window for **Line Segment**, locate the **Endpoint** section.

4 From the **Specify** list, choose **Coordinates**.

5 In the **x** text field, type `6.8[mm]`.

6 In the **y** text field, type `4.2[mm]`.

7 In the **z** text field, type `-sqrt((15[mm])^2-(4.2[mm])^2)`.

8 Locate the **Assigned Attributes** section. Select the **Construction geometry** checkbox.

Line Segment 2 (ls2)

1 Right-click **Line Segment 1 (ls1)** and choose **Duplicate**.

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

3 Click to select the **Activate Selection** toggle button for **Start vertex**.

4 In the tree, select **ext1**.

5 On the object **ext1**, select Point 9 only.

6 Locate the **Endpoint** section. In the **y** text field, type `-4.2[mm]`.

7 Click **Build Selected**.

Loft 1 (loft1)

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

2 In the **Settings** window for **Loft**, click to expand the **Start Profile** section.

3 Click to select the **Activate Selection** toggle button for **Start profile**.

4 On the object **dif1**, select Boundary 1 only.

5 Click to expand the **End Profile** section. Click to select the **Activate Selection** toggle button for **End profile**.

6 On the object **ext1**, select Boundary 8 only.

7 Click to expand the **Guide Curves** section. Click to select the **Activate Selection** toggle button for **Guide objects**.

8 Select the objects **ls1** and **ls2** only.

9 Click **Build Selected**.

Rigid Transform 1 (rt1)

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

2 Select the object **loft1** only.

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

4 In the **xw** text field, type `-50.5[mm]`.

5 Locate the **Rotation** section. In the **Angle** text field, type `180[deg]`.

6 Locate the **Input** section. Select the **Keep input objects** checkbox.

7 Click **Build Selected**.

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

Intersection 1 (int1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Intersection**.

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

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

4 Select the **Keep input objects** checkbox.

5 Click **Build Selected**.

Extract 1 (extract1)

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

2 In the **Settings** window for **Extract**, locate the **Entities or Objects to Extract** section.

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

4 On the object **int1**, select Edges 1, 6, 7, and 12 only.

5 From the **Input object handling** list, choose **Remove**.

6 Click **Build Selected**.

Copy 1 (copy1)

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

2 Select the object **extract1** only.

3 In the **Settings** window for **Copy**, click **Build Selected**.

Union 1 (uni1)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **extract1** and **loft1** only.

3 In the **Settings** window for **Union**, click **Build Selected**.

Union 2 (uni2)

1 In the **Geometry** toolbar, click **Booleans and Partitions** and choose **Union**.

2 Select the objects **copy1** and **rt1** only.

3 In the **Settings** window for **Union**, 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** checkbox.

Ignore Faces 1 (igf1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Faces**.

2 On the object **fin**, select Boundaries 6, 11, 34, and 39 only.

3 In the **Settings** window for **Ignore Faces**, locate the **Input** section.

4 Clear the **Ignore adjacent edges and vertices** checkbox.

5 Click **Build Selected**.

Ignore Edges 1 (ige1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Edges**.

2 On the object **igf1**, select Edges 12, 23, 79, and 85 only.

3 In the **Settings** window for **Ignore Edges**, click **Build Selected**.

Ignore Vertices 1 (igv1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Ignore Vertices**.

2 On the object **ige1**, select Points 9 and 54 only.

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

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

Convert to COMSOL 1 (ccom1)

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

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

Ignore Vertices 1 (igv1)

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

2 Click **Export**.

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

4 In the **Export[noun]** window for **Geometry**, locate the **Export** section.

5 In the **Filename** text field, type `contact_switch.mphbin`.

6 Click the **Export entire finalized geometry** button.

7 Click **Export**.

Convert to COMSOL 1 (ccom1)

In the **Model Builder** window, right-click **Convert to COMSOL 1 (ccom1)** and choose **Delete**.

Ignore Vertices 1 (igv1)

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