Created in COMSOL Multiphysics 5.6

# Contact Switch

## *Introduction*

A contact switch is used to regulate whether or not an electrical 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 electrical 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.

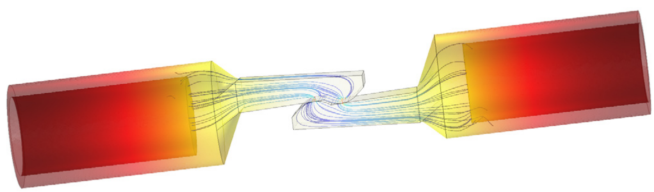

*Figure 1: A contact switch.*

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 this from happening. 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 at the contact surface.

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 for.

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.

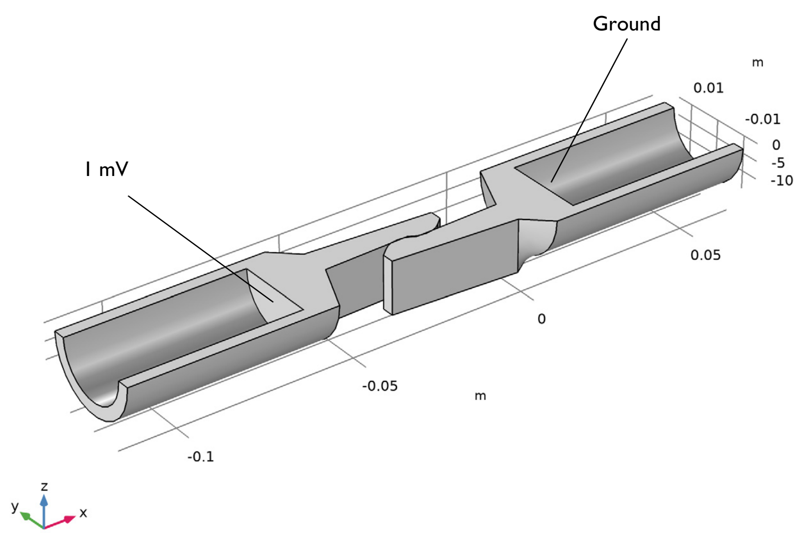


*Figure 2: Switch geometry.*

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.

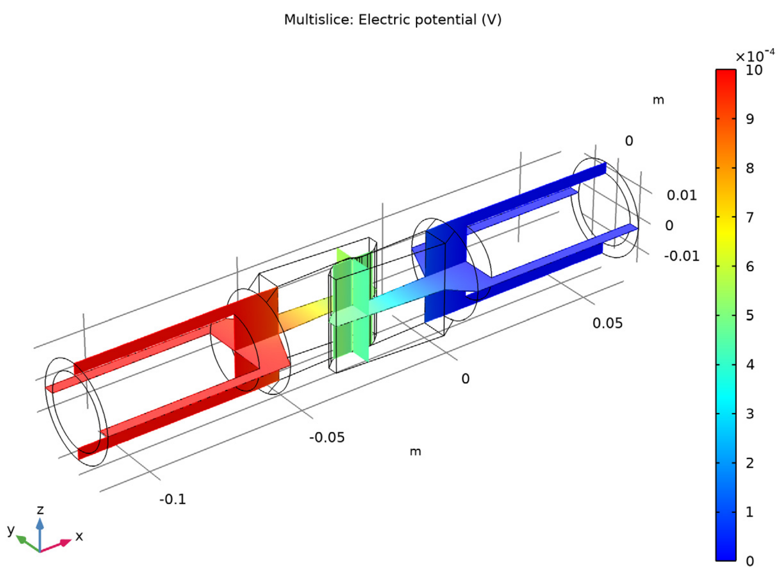


*Figure 3: Electric potential profile.*

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.

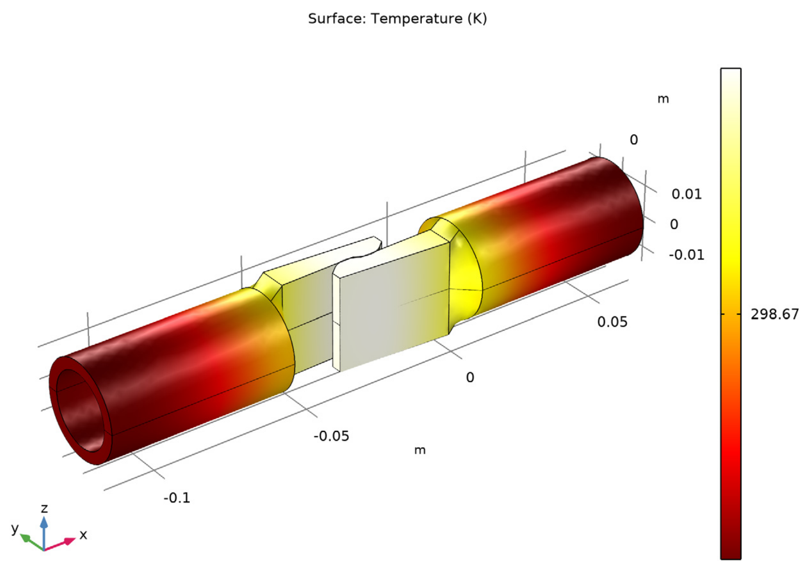


*Figure 4: Temperature distribution.*

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.

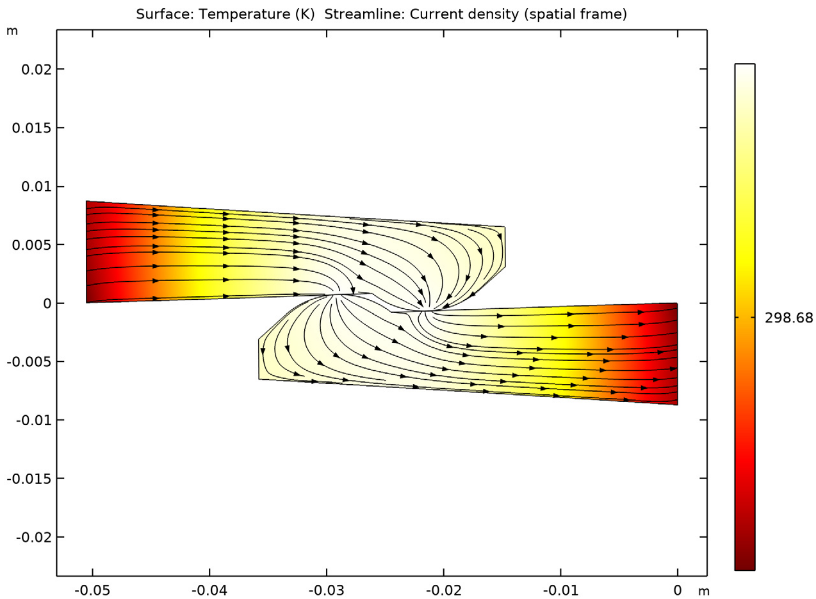


*Figure 5: Temperature distribution (volume plot) and current density (streamlines) at the contact region.*

---

**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

### *Import 1 (imp1)*

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 `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** check box.

6 In the **Home** 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 Select the **Activate Selection** toggle button.

5 Select Boundaries 25 and 28 only.

## ADD MATERIAL

1 In the **Home** 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 **Add to Component** in the window toolbar.

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

## SOLID MECHANICS (SOLID)

### *Fixed Constraint 1*

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Solid Mechanics (solid)** 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 **Physics** toolbar, click **Pairs** and choose **Contact**.

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

3 Under **Pairs**, click **Add**.

4 In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

5 Click **OK**.

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

7 From the **Formulation** list, choose **Augmented Lagrangian**.

8 Locate the **Initial Value** section. In the $T_\mathrm{n}$ text field, type `1e7`.

## 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 Click the **Convective heat flux** button.

**5** In the $h$ text field, type 2.

### *Symmetry 1*

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

**2** Select Boundaries 2 and 22 only.

### *Pair Thermal Contact 1*

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

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

**3** Under **Pairs**, click **Add**.

**4** In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

**5** Click **OK**.

**6** In the **Settings** window for **Pair Thermal Contact**, locate the **Contact Surface Properties** section.

**7** From the $p$ list, choose **Contact pressure (solid/cnt1)**.

## 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].

### *Pair Electrical Contact 1*

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

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

**3** Under **Pairs**, click **Add**.

**4** In the **Add** dialog box, select **Contact Pair 1 (p1)** in the **Pairs** list.

**5** Click **OK**.

**6** In the **Settings** window for **Pair Electrical Contact**, locate the **Contact Surface Properties** section.

7 From the $p$ list, choose **Contact pressure (solid/cnt1)**.

## 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. Select the **Maximum element size** check box.
7 In the associated text field, type `5e-4`.

### *Size*

1 In the **Model Builder** window, 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 table, clear the **Solve for** check boxes for **Electric Currents (ec)** and **Heat Transfer in Solids (ht)**.

### *Stationary 2*

1 In the **Study** toolbar, click **Study Steps** and choose **Stationary>Stationary**.
2 In the **Settings** window for **Stationary**, locate the **Physics and Variables Selection** section.
3 In the table, clear the **Solve for** check box 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 fourth default plots as in Figure 3 and Figure 4.

### *Study 1/Solution 1 (sol1)*

1 In the **Model Builder** window, expand the **Results>Datasets** node, then click **Study 1/ Solution 1 (sol1)**.

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

3 From the **Frame** list, choose **Spatial (x, y, z)**.

### *Study 1/Solution Store 1 (sol2)*

1 In the **Model Builder** window, click **Study 1/Solution Store 1 (sol2)**.

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

3 From the **Frame** list, choose **Spatial (x, y, z)**.

### *Mirror 3D 1*

1 In the **Results** toolbar, click **More Datasets** and choose **Mirror 3D**.

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

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

### *Electric Potential (ec)*

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

*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 **ThermalLight**.

*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 **Separating distance** text field, type `0.02`.

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**.