Heating Circuit

Introduction

Small heating circuits find use in many applications. For example, in manufacturing processes they heat up reactive fluids. Figure 1 illustrates a typical heating device for this application. The device consists of an electrically resistive layer deposited on a glass plate. The layer causes Joule heating when a voltage is applied to the circuit. The layer's properties determine the amount of heat produced.

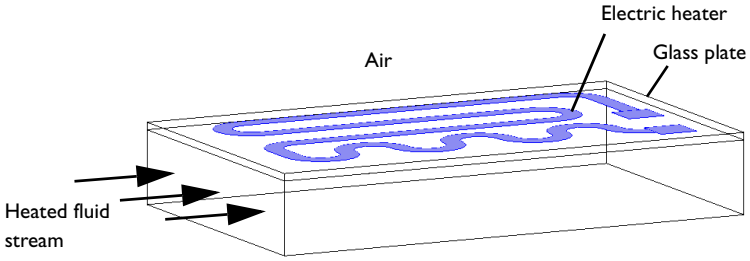


Figure 1: Geometry of a heating device.

In this particular application, you must observe three important design considerations:

- Noninvasive heating
- Minimal deflection of the heating device
- Avoidance of overheating the process fluid

The heater must also work without failure. You achieve the first and second requirements by inserting a glass plate between the heating circuit and the fluid; it acts as a conducting separator. Glass is an ideal material for both these purposes because it is nonreactive and has a low thermal-expansion coefficient.

You must also avoid overheating due to the risk of self-ignition of the reactive fluid stream. Ignition is also the main reason for separating the electrical circuit from direct contact with the fluid. The heating device is tailored for each application, making virtual prototyping very important for manufacturers.

For heating circuits in general, detachment of the resistive layer often determines the failure rate. This is caused by excessive thermally induced interfacial stresses. Once the layer has detached, it gets locally overheated, which accelerates the detachment. Finally, in the worst case, the circuit might overheat and burn. From this perspective,

©2012 COMSOL

it is also important to study the interfacial tension due to the different thermal-expansion coefficients of the resistive layer and the substrate as well as the differences in temperature. The geometric shape of the layer is a key parameter to design circuits that function properly. You can investigate all of the above-mentioned aspects by modeling the circuit.

This multiphysics example simulates the electrical heat generation, the heat transfer, and the mechanical stresses and deformations of a heating circuit device. The model uses the Heat Transfer interface of the Heat Transfer module in combination with the Shell, Conductive Media DC interface from the AC/DC Module and the Solid, Stress-Strain and Shell interfaces from the Structural Mechanics Module.

Note: This model requires the AC/DC Module, the Heat Transfer Module, and the Structural Mechanics Module.

Model Definition

Figure 2 shows a drawing of the modeled heating circuit.

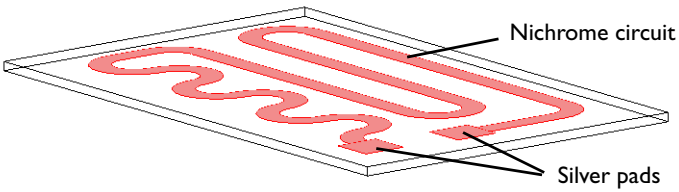


Figure 2: Drawing of the heating circuit deposited on a glass plate.

The device consists of a serpentine-shaped Nichrome resistive layer, 10 µm thick and 5 mm wide, deposited on a glass plate. At each end, it has a silver contact pad measuring 10 mm-by-10 mm-by-10 µm. When the circuit is in use, the deposited side of the glass plate is in contact with surrounding air, and the back side is in contact with the heated fluid. Assume that the edges of the glass plate are thermally insulated.

 ©2012 COMSOL

Table 1 gives the resistor's dimensions.

TABLE 1: DIMENSIONS

OBJECT	DIMENSION	SIZE
glass plate	length	130 mm
	width	80 mm
	thickness	2 mm
pads and circuit	thickness	10 μm

During operation the resistive layer produces heat. Model the electrically generated heat using the Shell, Conductive Media DC interface from the AC/DC Module. The governing equation is

$$\nabla_t \cdot (-d\sigma\nabla_t V) = 0$$

where d is the thin layer's thickness (m), σ is the electrical conductivity (S/m), V is the electric potential (V), and ∇_t denotes the gradient operator in the tangential directions. An actual applies 12 V to the pads. In the model you achieve this effect by setting the potential at one edge of the first pad to 12 V and that of one edge of the other pad to 0 V.

To model the heat transfer in the thin conducting layer, use the Highly Conductive Layer feature of the General Heat Transfer interface. It is then not necessary to add a separate interface for it.

The heat power per unit area (measured in W/m^2) produced inside the thin layer is given by

$$q_{\text{prod}} = d\,Q_{\text{DC}} \tag{1}$$

where $Q_{\text{DC}} = \mathbf{J} \cdot \mathbf{E} = \sigma|\nabla_t V|^2$ (W/m^3) is the power density. The generated heat appears as an inward heat flux at the surface of the glass plate.

At steady state, the resistive layer dissipates the heat it generates in two ways: on its up side to the surrounding air (at 293 K), and on its down side to the glass plate. The glass plate is similarly cooled in two ways: on its circuit side by air, and on its back side by a process fluid (353 K). You model the heat fluxes to the surroundings using heat transfer film coefficients, h. For the heat transfer to air, $h = 5$ $W/(m^2 \cdot K)$, representing natural convection. On the glass plate's back side, $h = 20$ $W/(m^2 \cdot K)$, representing convective heat transfer to the fluid. The sides of the glass plate are insulated.

The resulting heat transfer equation for the device, together with the boundary condition used to describe the heat fluxes at the front and back sides, is

$$\nabla \cdot (-k\nabla T) = 0$$

$$-\mathbf{n} \cdot (-k\nabla T) = q_0 + h(T_{\text{inf}} - T) - \nabla_t \cdot (-d_s k_s \nabla_t T)$$

where $\mathbf{n}$ is the normal vector of the boundary, k is the thermal conductivity (W/(m·K)), h is the heat transfer film coefficient (W/(m^2·K)), and T_{inf} is the temperature (K) of the surrounding medium. The last term on the right-hand side represents the additional flux given by the thin conducting layer, and the constant k_s is the thermal conductivity in the layer (W/(m·K)). This term is only present on the boundaries where the layer is present. Similarly, the inward heat flux, q_0, is equal to q_{prod} (see Equation 1) at the layer but vanishes elsewhere.

The model simulates thermal expansion using static structural-mechanics analyses. It uses the Solid, Stress-Strain interface for the glass plate, and the Shell interface for the circuit layer. The equations of these two interfaces are described in the *Structural Mechanics Module User's Guide*. The stresses are set to zero at 293 K. You determine the boundary conditions for the Solid, Stress-Strain interface by fixing one corner with respect to x-, y-, and z-displacements and rotation.

Table 2 summarizes the material properties used in the model.

TABLE 2: MATERIAL PROPERTIES

MATERIAL	E [GPa]	ν	α [ppm]	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silver	83	0.37	18.9	420	10500	230
Nichrome	213	0.33	10.0	15	9000	20
Glass	73.1	0.17	55	1.38	2203	703

 ©2012 COMSOL

Results and Discussion

Figure 3 shows the heat that the resistive layer generates.

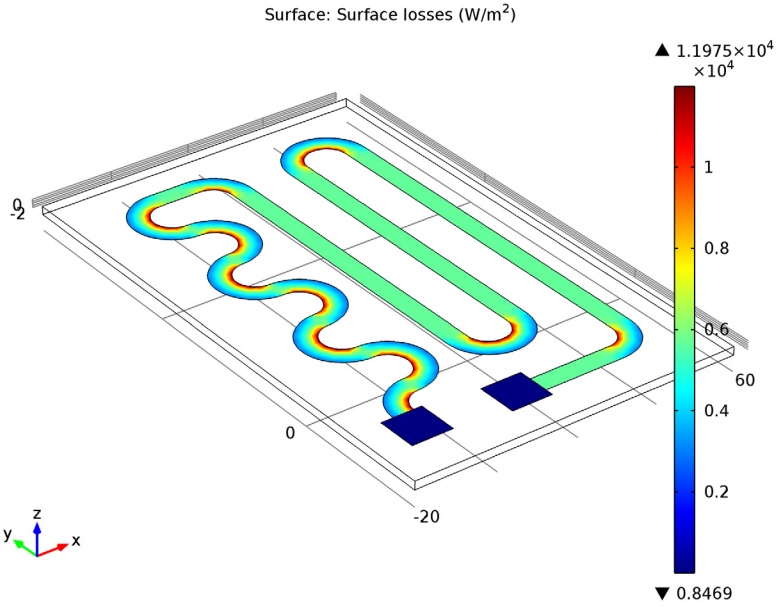


Figure 3: Stationary heat generation in the resistive layer when 12 V is applied.

The highest heating power arises in the inner corners of the curves due to the higher current density at these spots. The total generated heat, as calculated by integration, is approximately 13.8 W.

Figure 4 shows the temperature of the resistive layer and the glass plate at steady state.

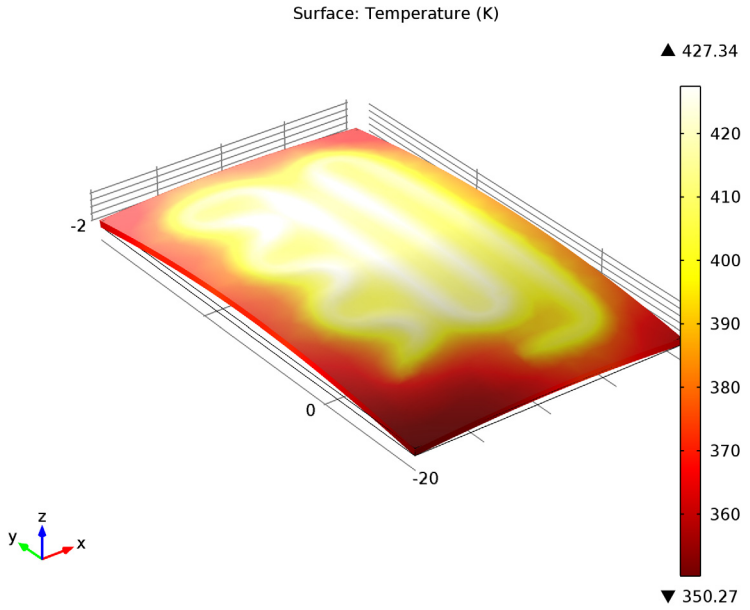


Figure 4: Temperature distribution in the heating device at steady state.

The highest temperature is approximately 430 K, and it appears in the central section of the circuit layer. It is interesting to see that the differences in temperature between the fluid side and the circuit side of the glass plate are quite small because the plate is very thin. Using boundary integration, the integral heat flux on the fluid side evaluates to approximately 8.5 W. This means that the device transfers the majority of the heat it generates—8.5 W out of 13.8 W—to the fluid, which is good from a design perspective, although the thermal resistance of the glass plate results in some losses.

The temperature rise also induces thermal stresses due the materials' different coefficients of thermal expansion. As a result, mechanical stresses and deformations arise in the layer and in the glass plate. Figure 5 shows the effective stress distribution

 ©2012 COMSOL

in the device and the resulting deformations. During operation, the glass plate bends towards the air side.

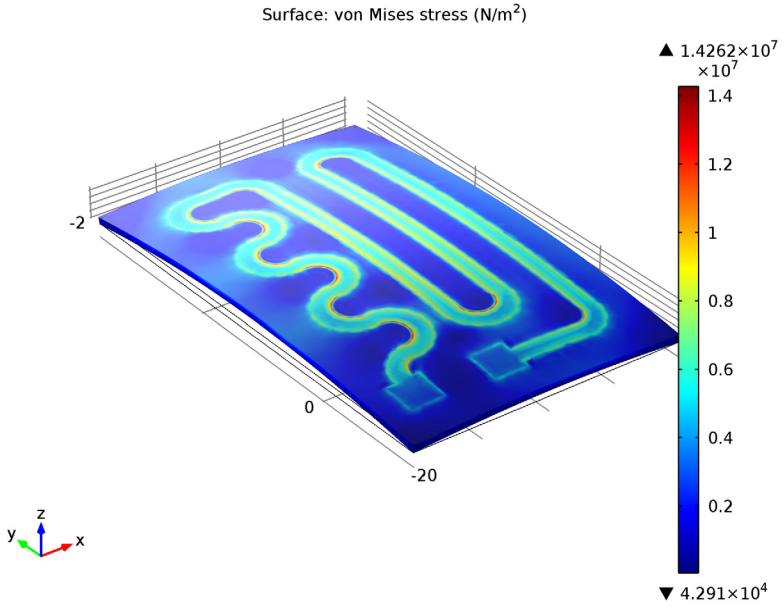


Figure 5: The thermally induced von Mises effective stress plotted with the deformation.

The highest effective stress, approximately 7 MPa, appears in the corners of the silver pads. The yield stress for high quality glass is roughly 250 MPa, and for Nichrome it is 360 MPa. This means that the individual objects remain structurally intact for the simulated heating power loads.

You must also consider stresses in the interface between the resistive layer and the glass plate. Assume that the yield stress of the surface adhesion in the interface is in the region of 50 MPa—a value significantly lower than the yield stresses of the other materials in the device. If the effective stress increases above this value, the resistive layer will locally detach from the glass. Once it has detached, heat transfer is locally impeded, which can lead to overheating of the resistive layer and eventually cause the device to fail.

Figure 6 displays the effective forces acting on the adhesive layer during heater operation. As the figure shows, the device experiences a maximum interfacial stress that is an order of magnitude smaller than the yield stress. This means that the device will be OK in terms of adhesive stress.

©2012 COMSOL

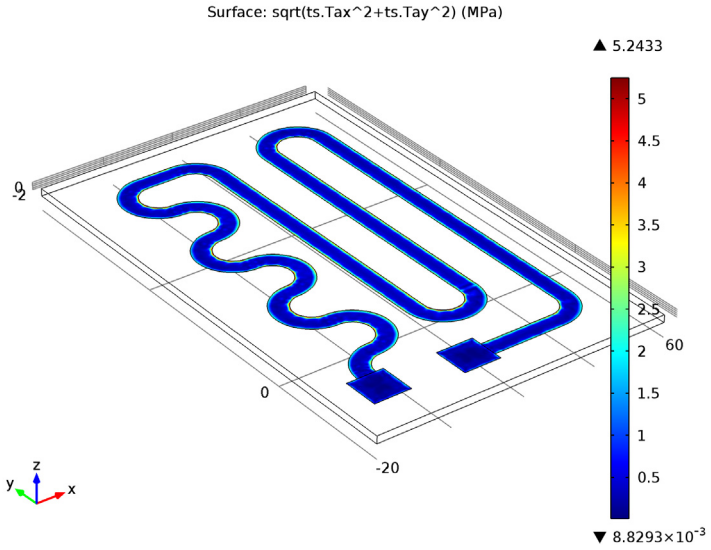


Figure 6: The effective forces in the interface between the resistive layer and the glass plate.

Finally study the device's deflections, depicted in Figure 7.

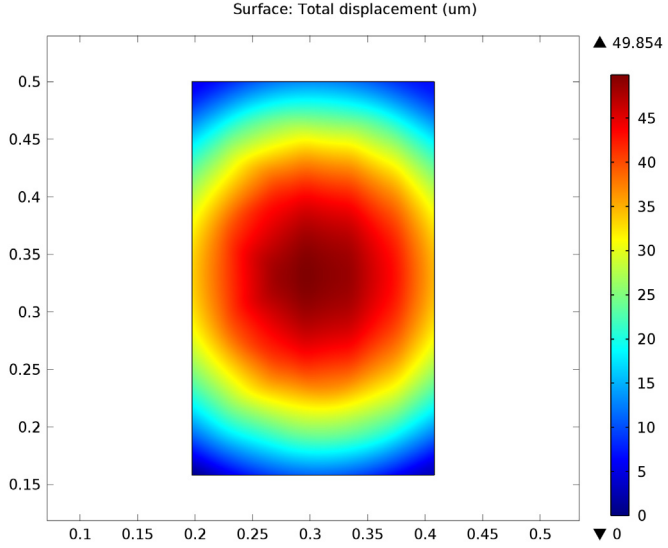


Figure 7: Total displacement on the fluid side of the glass plate.

 ©2012 COMSOL

The maximum displacement, located at the center of the plate, is approximately 30 μm. For high-precision applications, such as semiconductor processing, this might be a significant value that limits the device's operating temperature.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/heating_circuit`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Structural Mechanics>Thermal Stress (ts)**.

4 Click **Add Selected**.

5 In the **Add physics** tree, select **AC/DC>Electric Currents, Shell (ecs)**.

6 Click **Add Selected**.

7 In the **Add physics** tree, select **Structural Mechanics>Shell (shell)**.

8 Click **Add Selected**.

9 Click **Next**.

10 Find the **Studies** subsection. In the tree, select **Preset Studies for Selected Physics>Stationary**.

11 Click **Finish**.

The **Thermal Stress** physics solves for both the temperature and the displacement in the volume but just for the temperature in the shell representing the circuit. The **Electric Currents, Shell** physics models the electric potential in the shell representing the circuit, while the **Shell** physics solves for the displacement in the shell representing the circuit.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

Name	Expression	Description
V_in	12[V]	Input voltage
d_layer	10[um]	Layer thickness
sigma_silver	6.3e7[S/m]	Electric conductivity of silver
sigma_Nichrome	9.3e5[S/m]	Electric conductivity of Nichrome
T_air	20[degC]	Air temperature
h_air	5[W/(m^2*K)]	Heat transfer film coefficient, air
T_fluid	353[K]	Fluid temperature
h_fluid	20[W/(m^2*K)]	Heat transfer film coefficient, fluid

GEOMETRY 1

Start by creating the geometry of the heating circuit. To simplify this step, insert a prepared geometry sequence from file. After insertion you can study each geometry step in the sequence.

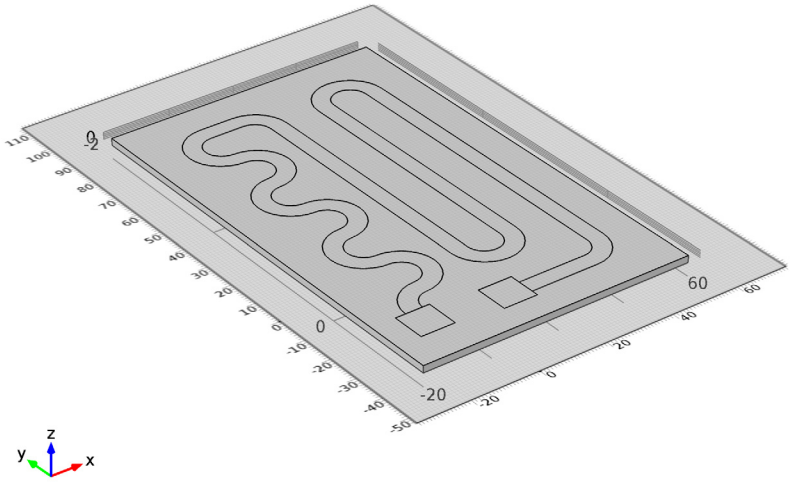
1 In the **Model Builder** window, under **Model 1** right-click **Geometry 1** and choose **Insert Sequence from File**.

2 Browse to the model's Model Library folder and double-click the file `heating_circuit_geom_sequence.mph`.

 ©2012 COMSOL

3 Click the **Build All** button.

The model geometry is now complete.



DEFINITIONS

Add a selection that you can use later when applying boundary conditions and shell physics settings.

Explicit 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.

2 In the **Explicit** settings window, locate the **Input Entities** section.

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

4 Select Boundaries 6–8 only.

5 Right-click **Model 1>Definitions>Explicit 1** and choose **Rename**.

6 Go to the **Rename Explicit** dialog box and type `Circuit` in the **New name** edit field.

7 Click **OK**.

Before creating the materials for use in this model, it is a good idea to specify which boundaries are to be modeled as conducting shells. Using this information, COMSOL Multiphysics can detect which material properties are needed.

THERMAL STRESS

Highly Conductive Layer 1

1 In the **Model Builder** window, under **Model 1** right-click **Thermal Stress** and choose the boundary condition **Heat Transfer>Highly Conductive Layer**.

2 In the **Highly Conductive Layer** settings window, locate the **Boundary Selection** section.

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

4 Locate the **Model Inputs** section. In the d_s edit field, type `d_layer`.

ELECTRIC CURRENTS, SHELL

1 In the **Model Builder** window, under **Model 1** click **Electric Currents, Shell**.

2 In the **Electric Currents, Shell** settings window, locate the **Boundary Selection** section.

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

4 Locate the **Shell Thickness** section. In the d_s edit field, type `d_layer`.

SHELL

1 In the **Model Builder** window, under **Model 1** click **Shell**.

2 In the **Shell** settings window, locate the **Boundary Selection** section.

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

4 Locate the **Thickness** section. In the d edit field, type `d_layer`.

5 Click to expand the **Dependent Variables** section. In the **Displacement field** edit field, type `u`.

Linear Elastic Material 1

In the **Model Builder** window, expand the **Shell** node.

Thermal Expansion 1

1 Right-click **Linear Elastic Material 1** and choose **Thermal Expansion**.

2 In the **Thermal Expansion** settings window, locate the **Model Inputs** section.

3 From the T list, choose **Temperature (ts/tle1)**.

 ©2012 COMSOL

MATERIALS

Material Browser

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

3 In the tree, select **Built-In>Silica glass**.

4 Right-click and choose **Add Material to Model** from the menu.

Now set up the materials.

Material 2

1 In the **Model Builder** window, right-click **Materials** and choose **Material**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Circuit**.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Thermal conductivity	k	420
Density	rho	10500
Heat capacity at constant pressure	Cp	230
Relative permittivity	epsilonr	1
Electrical conductivity	sigma	sigma_silver
Young's modulus	E	83e9
Poisson's ratio	nu	0.37
Coefficient of thermal expansion	alpha	18.9e-6

6 In the **Model Builder** window, right-click **Material 2** and choose **Rename**.

7 Go to the **Rename Material** dialog box and type `Silver` in the **New name** edit field.

8 Click **OK**.

Material 3

1 Right-click **Materials** and choose **Material**.

2 In the **Material** settings window, locate the **Geometric Entity Selection** section.

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

4 Select Boundary 7 only.

5 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value
Thermal conductivity	k	15
Density	rho	9000
Heat capacity at constant pressure	Cp	20
Relative permittivity	epsilonr	1
Electrical conductivity	sigma	sigma_Nichrome
Young's modulus	E	213e9
Poisson's ratio	nu	0.33
Coefficient of thermal expansion	alpha	10e-6

6 In the **Model Builder** window, right-click **Material 3** and choose **Rename**.

7 Go to the **Rename Material** dialog box and type `Nichrome` in the **New name** edit field.

8 Click **OK**.

With the materials defined, set up the remaining physics of the model. In the next section, the resistive loss within the circuit is defined as a heat source for the thermal stress physics. The resistive loss is calculated automatically within the Electric Currents, Shell physics. You can find out the variable name by clicking the **Show** button in the **Model Builder** window's toolbar and activating **Equation View**. In the **Electric Currents, Shell > Current Conservation 1 > Equation View** window, locate the **Variables** section. Here you will find the variable `ecs.Qsh`, which represents the surface losses and is basically calculate J^2E.

THERMAL STRESS

Boundary Heat Source 1

1 In the **Model Builder** window, under **Model 1** right-click **Thermal Stress** and choose the boundary condition **Heat Transfer>Boundary Heat Source**.

2 In the **Boundary Heat Source** settings window, locate the **Boundary Selection** section.

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

4 Locate the **Boundary Heat Source** section. In the Q_b edit field, type `ecs.Qsh`.

Convective Cooling 1

1 In the **Model Builder** window, right-click **Thermal Stress** and choose the boundary condition **Heat Transfer>Convective Cooling**.

2 Select Boundaries 4 and 6–8 only.

 ©2012 COMSOL

3 In the **Convective Cooling** settings window, locate the **Heat Flux** section.

4 In the h edit field, type `h_air`.

5 In the T_{ext} edit field, type `T_air`.

Convective Cooling 2

1 Right-click **Thermal Stress** and choose the boundary condition **Heat Transfer>Convective Cooling**.

2 Select Boundary 3 only.

3 In the **Convective Cooling** settings window, locate the **Heat Flux** section.

4 In the h edit field, type `h_fluid`.

5 In the T_{ext} edit field, type `T_fluid`.

Next, add constraints to restrain the glass plate movements.

Fixed Constraint 1

1 Right-click **Thermal Stress** and choose **Points>Fixed Constraint**.

2 Select Point 1 only.

Prescribed Displacement 1

1 Right-click **Thermal Stress** and choose **Points>Prescribed Displacement**.

2 Select Point 63 only.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in y direction** check box.

5 Select the **Prescribed in z direction** check box.

Prescribed Displacement 2

1 Right-click **Thermal Stress** and choose **Points>Prescribed Displacement**.

2 Select Point 3 only.

3 In the **Prescribed Displacement** settings window, locate the **Prescribed Displacement** section.

4 Select the **Prescribed in z direction** check box.

Finally, add a voltage and ground.

ELECTRIC CURRENTS, SHELL

Electric Potential 1

1 In the **Model Builder** window, under **Model 1** right-click **Electric Currents, Shell** and choose **Electric Potential**.

2 Select Edge 10 only.

3 In the **Electric Potential** settings window, locate the **Electric Potential** section.

4 In the V_0 edit field, type `V_in`.

Ground 1

1 In the **Model Builder** window, right-click **Electric Currents, Shell** and choose **Ground**.

2 Select Edge 43 only.

MESH 1

Use a relatively coarse mesh to reduce the model size.

Free Triangular 1

1 In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **More Operations>Free Triangular**.

2 Select Boundaries 4 and 6–8 only.

Size 1

1 Right-click **Model 1>Mesh 1>Free Triangular 1** and choose **Size**.

2 In the **Size** settings window, locate the **Geometric Entity Selection** section.

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

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

5 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.

6 In the associated edit field, type `4`.

Swept 1

In the **Model Builder** window, right-click **Mesh 1** and choose **Swept**.

Distribution 1

1 In the **Model Builder** window, under **Model 1>Mesh 1** right-click **Swept 1** and choose **Distribution**.

2 In the **Distribution** settings window, locate the **Distribution** section.

3 In the **Number of elements** edit field, type `2`.

 ©2012 COMSOL

4 Click the **Build All** button.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

The default plots show the von Mises stress (Figure 5) and the temperature (Figure 4) on the surface of the full 3D geometry including the deformation, the electric potential on the circuit layer, and the von Mises stress on the top and bottom of the circuit layer.

To generate Figure 3 follow the steps below.

3D Plot Group 6

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

2 Right-click **3D Plot Group 6** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electric Currents, Shell>Heating and losses>Surface losses (ecs.Qsh)**.

4 Click the **Plot** button.

5 Click the **Scene Light** button on the Graphics toolbar.

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

7 In the **Model Builder** window, right-click **3D Plot Group 6** and choose **Rename**.

8 Go to the **Rename 3D Plot Group** dialog box and type `Surface Losses` in the **New name** edit field.

9 Click **OK**.

The following steps generate a plot of the norm of the surface traction vector in the surface plane (see Figure 6):

Data Sets

1 In the **Model Builder** window, under **Results>Data Sets** right-click **Solution 1** and choose **Duplicate**.

2 Right-click **Results>Data Sets>Solution 2** and choose **Add Selection**.

3 In the **Selection** settings window, locate the **Geometric Entity Selection** section.

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

5 From the **Selection** list, choose **Circuit**.

3D Plot Group 7

1 In the **Model Builder** window, right-click **Results** and choose **3D Plot Group**.

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

3 From the **Data set** list, choose **Solution 2**.

4 Right-click **Results>3D Plot Group 7** and choose **Surface**.

5 In the **Surface** settings window, locate the **Expression** section.

6 In the **Expression** edit field, type `sqrt(ts.Tax^2+ts.Tay^2)`.

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

8 Click the **Plot** button.

9 In the **Model Builder** window, right-click **3D Plot Group 7** and choose **Rename**.

10 Go to the **Rename 3D Plot Group** dialog box and type `Interface Stress` in the **New name** edit field.

11 Click **OK**.

Finally, to obtain Figure 7, proceed as follows:

Data Sets

1 In the **Model Builder** window, under **Results** right-click **Data Sets** and choose **Surface**.

2 Select Boundary 3 only.

2D Plot Group 8

1 In the **Model Builder** window, right-click **Results** and choose **2D Plot Group**.

2 Right-click **2D Plot Group 8** and choose **Surface**.

3 In the **Surface** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Thermal Stress (Solid Mechanics)>Displacement>Total displacement (ts.disp)**.

4 Locate the **Expression** section. From the **Unit** list, choose **um**.

5 Click the **Plot** button.

6 In the **Model Builder** window, right-click **2D Plot Group 8** and choose **Rename**.

7 Go to the **Rename 2D Plot Group** dialog box and type `Displacement, bottom boundary` in the **New name** edit field.

8 Click **OK**.

Derived Values

To calculate the values for the total generated heat and the integrated heat flux on the fluid side, perform a boundary integration:

 ©2012 COMSOL

1 In the **Model Builder** window, under **Results** right-click **Derived Values** and choose **Integration>Surface Integration**.

2 Select Boundary 3 only.

3 In the **Surface Integration** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Thermal Stress (Heat Transfer)>Convective heat flux (ts.ccflux)**.

4 Click the **Evaluate** button.

The result should be close to 13.8 W.

5 In the **Model Builder** window, right-click **Derived Values** and choose **Integration>Surface Integration**.

6 In the **Surface Integration** settings window, locate the **Selection** section.

7 From the **Selection** list, choose **Circuit**.

8 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Electric Currents, Shell>Heating and losses>Surface losses (ecs.Qsh)**.

9 Click the **Evaluate** button.

The result should be close to 8.5 W.

 ©2012 COMSOL