Created in COMSOL Multiphysics 6.2

Heating Circuit — Layered Shell Version

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 shown in this model. 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.

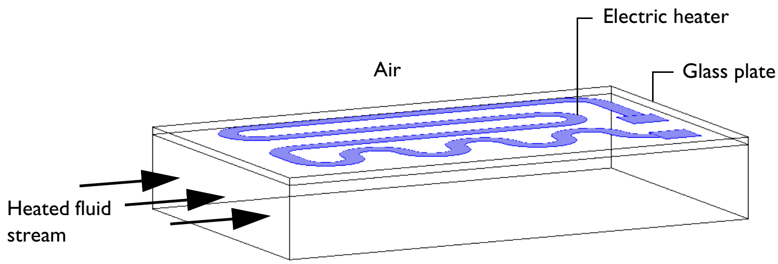


Figure 1: Geometry of a heating device.

In this particular model, there are 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. The first and second requirements are achieved 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 coefficient of thermal expansion.

Overheating must be avoided 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. Heating devices are tailored for each application, making virtual prototyping very important for manufacturers.

For heating circuits in general, the 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 further accelerates the detachment. Finally, in the worst case, the circuit might overheat and burn. From this perspective, 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 for proper functioning. You can investigate all of the abovementioned 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 in Shells interface of the Heat Transfer Module in combination with the Electric Currents, Layered Shell interface from the AC/DC Module or the MEMS Module, and the Layered Shell interface from the Composite Materials Module.

Note: In addition to the Composite Materials Module and the Structural Mechanics Module, this model requires the Heat Transfer Module and either the AC/DC Module or the MEMS Module. It is a layered shell version of the model *Heating Circuit* available in the AC/DC Module, Heat Transfer Module, and Structural Mechanics Module Application Libraries.

Model Definition

Figure 2 shows a drawing of the modeled heating circuit.

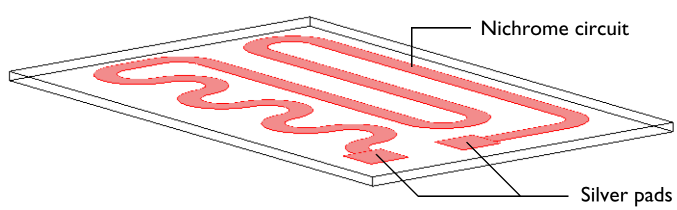


Figure 2: Drawing of the heating circuit deposited on a glass plate.

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 and sides of the glass plate are thermally insulated.

Table 1 lists the resistor's dimensions.

TABLE 1: DIMENSIONS.

OBJECT	LENGTH	WIDTH	THICKNESS
Glass plate	130 mm	80 mm	2 mm
Pads and circuit	-	-	10 µm

LAYERED SHELL APPROACH

Since this model uses a layered shell interface, in which the through thickness integration is inherent to the layered shell formulation itself, a surface geometry as shown in the Figure 3 is sufficient.

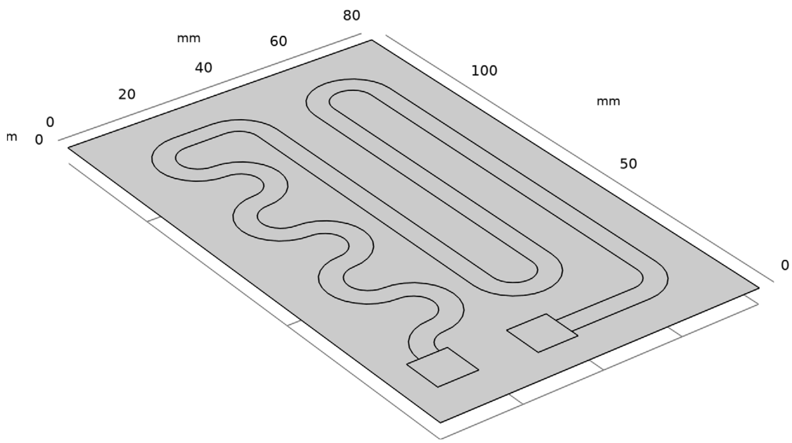


Figure 3: Layered shell version of the model geometry.

The layered shell geometry has many zones with layers of different materials and thicknesses. The spatial position of the different zones can be seen in Figure 4 and material and thicknesses for each zone can be seen in Figure 5.

The different zones of the layered shell are by default disconnected. Therefore, continuity conditions connecting layers from neighboring zones are required in all physics interfaces.

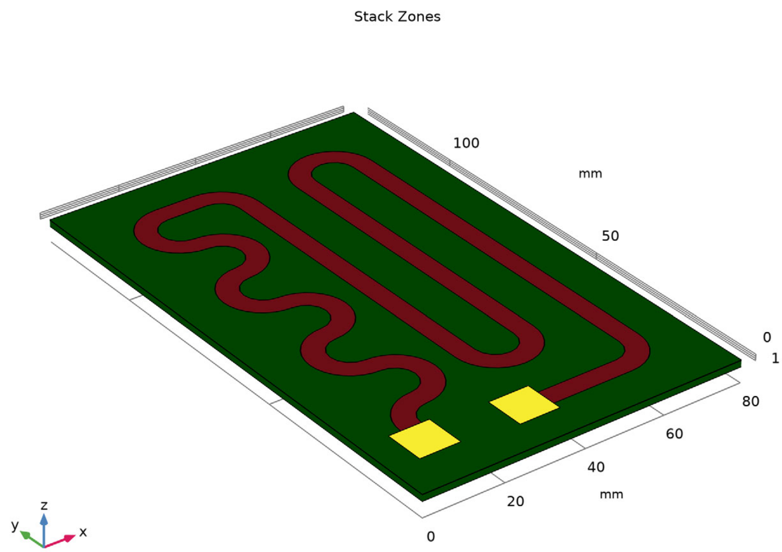


Figure 4: Zones with different layers in the layered shell geometry. The green zone has only glass layer, the red zone has glass and Nichrome layers, and yellow zone has glass and Silver layers.

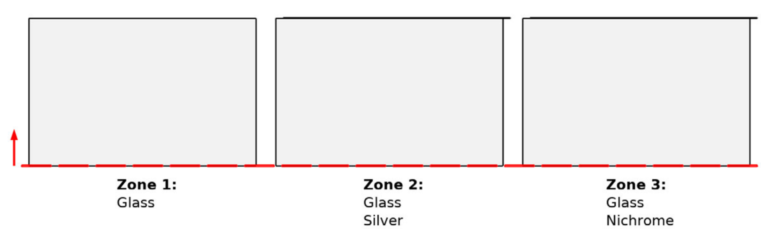


Figure 5: Cross-sectional view of zones having layers of different material and thickness.

BOUNDARY CONDITIONS

During operation the resistive layer produces heat. Model the electrically generated heat using the Electric Currents, Layered Shell interface from the AC/DC Module or the MEMS Module. An electric potential of 12 V is applied 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 Heat Transfer in Shells interface. The heat rate per unit area (measured in W/m^2) produced inside the thin layer is given by

$$q_{\mathrm{prod}} = dQ_{\mathrm{DC}} \tag{1}$$

where $Q_{\mathrm{DC}} = \mathbf{J} \cdot \mathbf{E} = \sigma|\nabla_{\mathbf{t}} V|^2$ (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 upside to the surrounding air (at 293 K), and on its downside 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 coefficients, h. For the heat transfer to air, $h = 5$ W/(m^2·K), representing natural convection. On the glass plate's back side, $h = 20$ W/(m^2·K), representing convective heat transfer to the fluid. The sides of the glass plate are insulated.

The model simulates thermal expansion using static structural mechanics analyses. It uses the Layered Shell interface for the glass plate as well as for the circuit layer. The stresses are set to zero at 293 K. You determine the boundary conditions for the Layered Shell interface by adding a rigid motion suppression node and restricting all rigid body modes.

MATERIAL PROPERTIES

Table 2 summarizes the material properties used in the model.

TABLE 2: MATERIAL PROPERTIES.

MATERIAL	E [GPa]	ν	α [1/K]	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silver	83	0.37	1.89e-5	420	10500	230
Nichrome	213	0.33	1e-5	15	9000	20
Glass	73.1	0.17	5.5e-7	1.38	2203	703

Results and Discussion

Figure 6 shows the heat that the resistive layer generates.

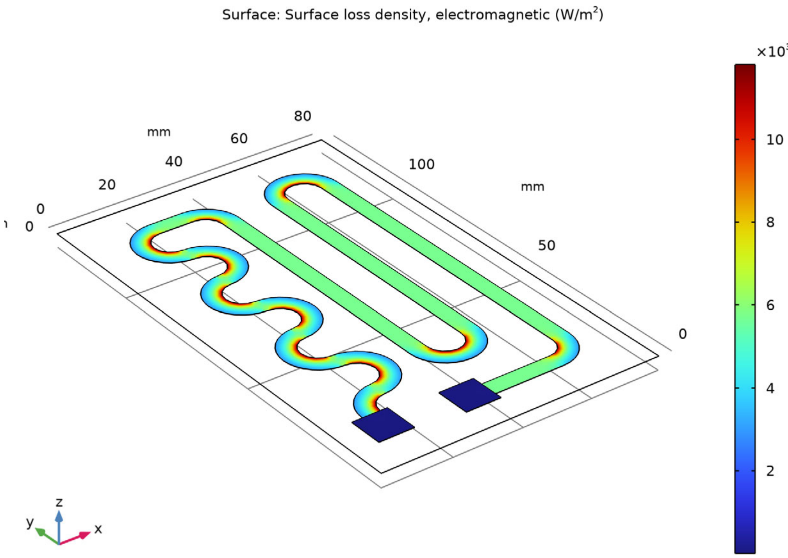


Figure 6: Stationary heat generation in the resistive layer when 12 V is applied.

The highest heating power occurs at 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 7 shows the temperature of the resistive layer and the glass plate at steady state.

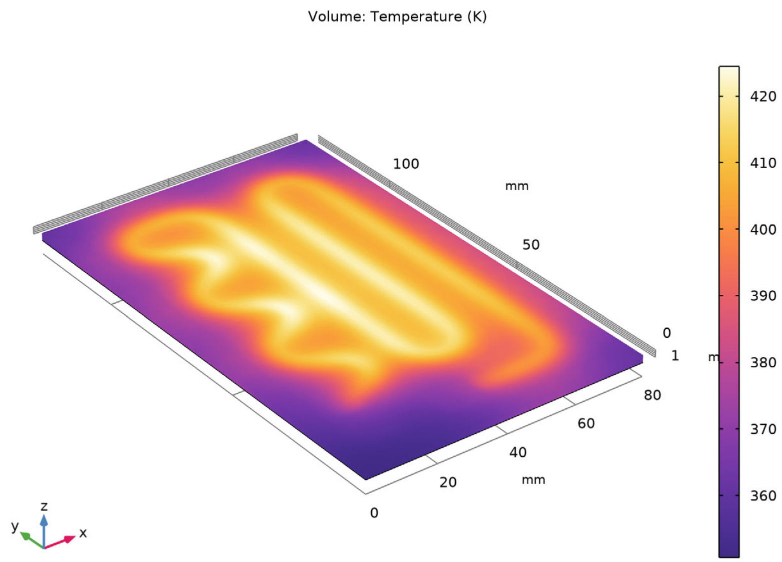


Figure 7: Temperature distribution in the heating device at steady state.

The highest temperature is approximately 428 K, and it appears in the central section of the circuit layer. Interestingly, the temperature differences 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 8 and Figure 9 shows the effective stress distribution in the device and the resulting deformations. During operation, the glass plate bends toward the air side.

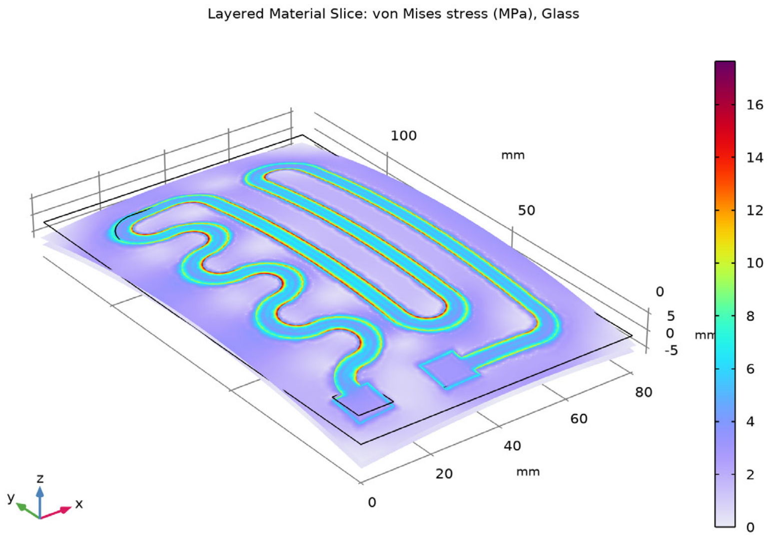


Figure 8: The thermally induced von Mises stress plotted with the deformation in glass plate.

The highest effective stress, approximately 26 MPa, occurs at the inner corners of the curves of the Nichrome circuit. 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.

Stresses in the interface between the resistive layer and the glass plate must also be considered. 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 locally detaches 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.

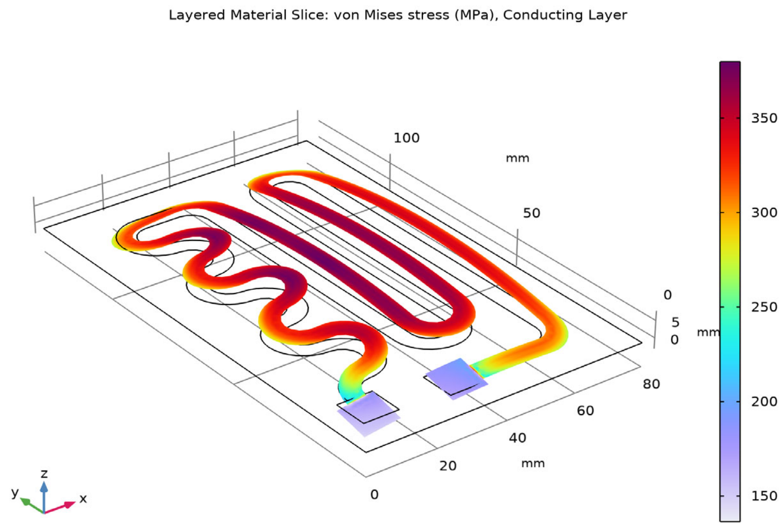


Figure 9: The thermally induced von Mises stress plotted with the deformation in resistive layer.

Figure 10 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 approximately five times smaller than the yield stress. This means that the device is safe in terms of the adhesive stress.

Finally, the warping of the device, that is, its deviation from a plane surface as shown in Figure 11 are studied. The maximum deviation from being a planar surface, is approximately 50 μm. For high-precision applications, such as semiconductor processing, this might be a significant value that limits the device's operating temperature.

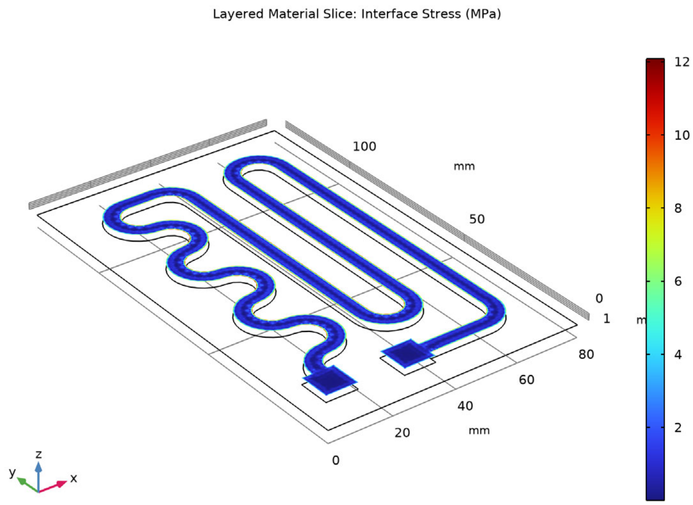


Figure 10: The effective forces in the interface between the resistive layer and the glass plate.

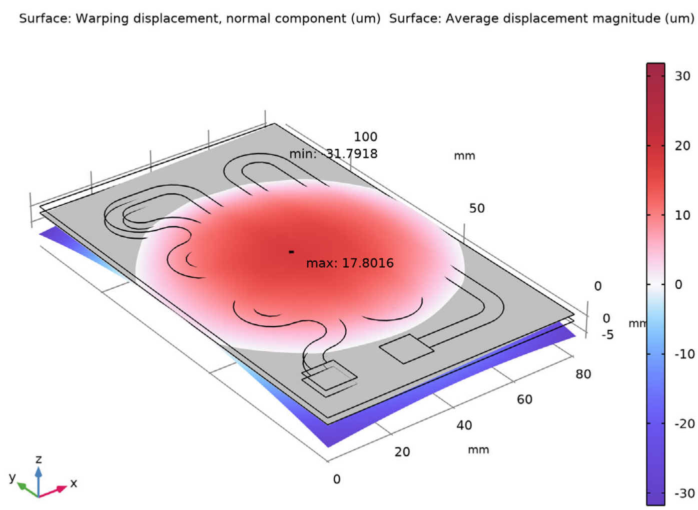


Figure 11: Deviation from a plane surface on the fluid side of the glass plate.

Application Library path: `Composite_Materials_Module/Multiphysics/heating_circuit_layered`

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>Thermal–Structure Interaction>Thermal Stress, Layered Shell**.
3 Click **Add**.
4 In the **Select Physics** tree, select **AC/DC>Electric Fields and Currents>Electric Currents in Layered Shells (ecis)**.
5 Click **Add**.
6 Click **Study**.
7 In the **Select Study** tree, select **General Studies>Stationary**.
8 Click **Done**.

The **Thermal Stress, Layered Shell** interface includes the **Heat Transfer in Shells** and the **Layered Shell** interfaces. In the silica glass, these two interfaces solve for temperature and displacements, respectively. In the conducting layer representing the circuit, the temperature, electrical potential, and displacement are solved by **Heat Transfer In Shells**, **Electric Currents in Layered Shell**, and the **Layered Shell** interfaces.

GLOBAL DEFINITIONS

Parameters 1

Load the parameters from a file.

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.
2 In the **Settings** window for **Parameters**, locate the **Parameters** section.
3 Click **Load from File**.

4 Browse to the model's Application Libraries folder and double-click the file `heating_circuit_layered_parameters.txt`.

If you do not want to build the geometry manually, you can load the geometry sequence from the stored model. In the **Model Builder** window, under **Component 1 (comp1)** right-click **Geometry 1** and choose **Insert Sequence**. Browse to the model's Application Libraries folder and double-click the file `heating_circuit_layered.mph`. You can then continue to the **Definitions** section below.

To build the geometry from scratch, continue here.

GEOMETRY 1

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

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

3 From the **Length unit** list, choose **mm**.

Work Plane 1 (wp1)

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

2 In the **Settings** window for **Work Plane**, click **Go to Plane Geometry**.

Work Plane 1 (wp1)>Plane Geometry

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

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

Work Plane 1 (wp1)>Square 1 (sq1)

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

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

3 In the **Side length** text field, type `10`.

4 Locate the **Position** section. In the **xw** text field, type `7`.

5 In the **yw** text field, type `10`.

6 Click **Build Selected**.

7 Right-click **Square 1 (sq1)** and choose **Duplicate**.

Work Plane 1 (wp1)>Square 2 (sq2)

1 In the **Model Builder** window, click **Square 2 (sq2)**.

2 In the **Settings** window for **Square**, locate the **Position** section.

3 In the **xw** text field, type `30`.

4 In the **yw** text field, type `8`.

5 Click **Build Selected**.

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 From the **Data source** list, choose **File**.

4 Click **Browse**.

5 Browse to the model's Application Libraries folder and double-click the file `heating_circuit_layered_polygon.txt`.

6 Click **Build Selected**.

Work Plane 1 (wp1)>Fillet 1 (fil1)

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

2 On the object **pol1**, select Points 2–8, 23–29, 34, 36, 37, 41, and 42 only.

   It might be easier to select the points by using the **Selection List** window. To open this window, navigate to the **Home** toolbar, click **Windows**, and choose **Selection List**. (If you are running the cross-platform desktop, you find **Windows** in the main menu.)

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `10`.

5 Click **Build Selected**.

Work Plane 1 (wp1)>Fillet 2 (fil2)

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

2 On the object **fil1**, select Points 6–12, 26–31, 37, 40, 43, 46, 49, and 50 only.

3 In the **Settings** window for **Fillet**, locate the **Radius** section.

4 In the **Radius** text field, type `5`.

5 Click **Build Selected**.

Work Plane 1 (wp1)>Rectangle 1 (r1)

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

2 In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.

3 In the **Width** text field, type `80`.

4 In the **Height** text field, type `130`.

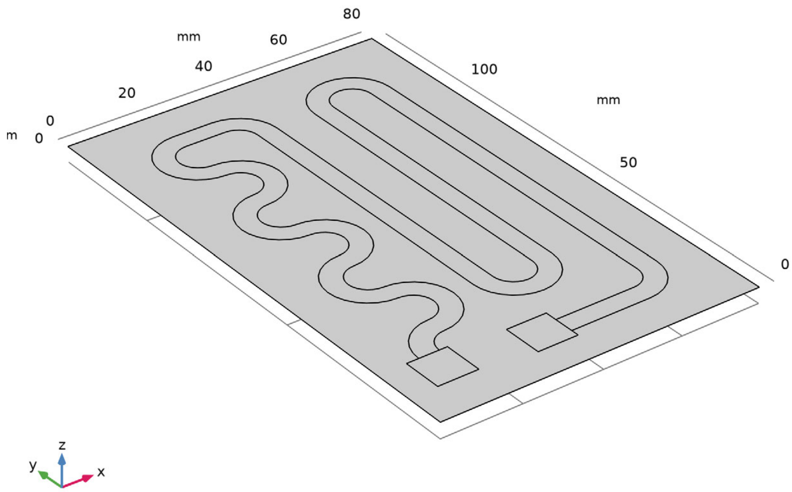
5 In the **Work Plane** toolbar, click **Build All**.

Form Union (fin)

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

The geometry should look like the figure below.

2 In the **Model Builder** window, under **Component 1 (comp1)>Geometry 1** click **Form Union (fin)**.



Before creating layered material stacks, add the materials for individual layers.

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>Silica glass**.

4 Right-click and choose **Add to Global Materials**.

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

GLOBAL DEFINITIONS

Silica Glass

In the **Settings** window for **Material**, type `Silica Glass` in the **Label** text field.

Silver Layer

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

2 Right-click **Material 2 (mat2)** and choose **Rename**.

3 In the **Rename Material** dialog box, type `Silver Layer` in the **New label** text field.

4 Click **OK**.

Nichrome Layer

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

2 Right-click **Material 3 (mat3)** and choose **Rename**.

3 In the **Rename Material** dialog box, type `Nichrome Layer` in the **New label** text field.

4 Click **OK**.

The stacking of the Silver and Nichrome layers is not the same across the glass plate. In order to model the stacking, add a **Layered Material Stack** node with **Layered Material** subnodes having different selections.

MATERIALS

Layered Material Stack 1 (stlmat1)

In the **Model Builder** window, under **Component 1 (comp1)** right-click **Materials** and choose **Layers>Layered Material Stack**.

Layered Material Link 1 (stlmat1.stllmat1)

In the **Model Builder** window, right-click **Layered Material Link 1 (stlmat1.stllmat1)** and choose **Delete**.

Layered Material Stack 1 (stlmat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Layered Material Stack 1 (stlmat1)**.

2 In the **Settings** window for **Layered Material Stack**, locate the **Orientation and Position** section.

3 From the **Position** list, choose **Bottom side on boundary**.

Glass

1 Right-click **Layered Material Stack 1 (stlmat1)** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Glass` in the **Label** text field.

3 Locate the **Layer Definition** section. In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness (m)	Mesh elements
`Layer 1`	Silica Glass (mat1)	`0.0`	`d_glass`	`3`

Silver

1 Right-click **Layered Material Stack 1 (stlmat1)** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Silver` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `2 4` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.

8 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness (m)	Mesh elements
`Layer 1`	Silver Layer (mat2)	`0.0`	`d_layer`	`1`

Nichrome

1 Right-click **Layered Material Stack 1 (stlmat1)** and choose **Layered Material**.

2 In the **Settings** window for **Layered Material**, type `Nichrome` in the **Label** text field.

3 Locate the **Boundary Selection** section. Click **Clear Selection**.

4 Click **Paste Selection**.

5 In the **Paste Selection** dialog box, type `3` in the **Selection** text field.

6 Click **OK**.

7 In the **Settings** window for **Layered Material**, locate the **Layer Definition** section.

8 In the table, enter the following settings:

Layer	Material	Rotation (deg)	Thickness (m)	Mesh elements
`Layer 1`	Nichrome Layer (mat3)	`0.0`	`d_layer`	`1`

To visualize the stacking, create a **Layer Cross Section Preview** plot through an action button in the **Layered Material Settings** section.

Layered Material Stack 1 (stlmat1)

1 In the **Model Builder** window, click **Layered Material Stack 1 (stlmat1)**.

2 In the **Settings** window for **Layered Material Stack**, click **Layer Cross-Section Preview** in the upper-right corner of the **Layered Material Settings** section. From the menu, choose **Create Layer Cross-Section Plot**.

RESULTS

Layer Cross-Section Preview

1 In the **Model Builder** window, expand the **Results** node, then click **Layer Cross-Section Preview**.

2 In the **Layer Cross-Section Preview** toolbar, click **Plot**.

Before adding the material properties, it is a good idea to first set up the physics, so that COMSOL Multiphysics can detect which material properties are needed.

LAYERED SHELL (LSHELL)

Linear Elastic Material 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Layered Shell (lshell)** click **Linear Elastic Material 1**.

2 In the **Settings** window for **Linear Elastic Material**, locate the **Linear Elastic Material** section.

3 From the **Material symmetry** list, choose **Isotropic**.

Rigid Motion Suppression 1

In the **Physics** toolbar, click **Boundaries** and choose **Rigid Motion Suppression**.

Continuity 1

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone1)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

Continuity 2

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone1)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone3)**.

Continuity 3

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone3)**.

5 In the **Selection** table, enter the following settings:

	Layered material	**Offset (m)**
√	Nichrome	0

The absolute displacement of the glass plate is not important in itself, since it is just a function of how the rigid body constraints are applied. Instead, you want to see how much the boundary deviates from being planar.

Warpage 1

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

2 In the **Settings** window for **Warpage**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Interface Selection** section. From the **Apply to** list, choose **Bottom interface**.

HEAT TRANSFER IN SHELLS (HTLSH)

Solid 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Heat Transfer in Shells (htlsh)** click **Solid 1**.

2 In the **Settings** window for **Solid**, locate the **Layer Model** section.

3 Clear the **Layerwise constant properties** check box.

Heat Flux, Interface 1

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

2 In the **Settings** window for **Heat Flux, Interface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Interface Selection** section. From the **Apply to** list, choose **Top interface**.

5 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

6 In the h text field, type `h_air`.

7 In the T_{ext} text field, type `T_air`.

Heat Flux, Interface 2

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

2 In the **Settings** window for **Heat Flux, Interface**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

4 Locate the **Interface Selection** section. From the **Apply to** list, choose **Bottom interface**.

5 Locate the **Heat Flux** section. From the **Flux type** list, choose **Convective heat flux**.

6 In the h text field, type `h_fluid`.

7 In the T_{ext} text field, type `T_fluid`.

Continuity 1

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone1)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

Continuity 2

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone1)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone3)**.

Continuity 3

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone3)**.

5 In the **Selection** table, enter the following settings:

	Layered material	Offset (m)
√	Nichrome	0

ELECTRIC CURRENTS IN LAYERED SHELLS (ECIS)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Electric Currents in Layered Shells (ecis)**.

2 In the **Settings** window for **Electric Currents in Layered Shells**, locate the **Boundary Selection** section.

3 In the list, select **1 (stlmat1)**.

4 Click **Remove from Selection**.

5 Select Boundaries 2–4 only.

6 Locate the **Shell Properties** section. Clear the **Use all layers** check box.

7 In the **Selection** table, clear the check box for **Layer 1 - Glass**.

Ground 1

1 In the **Physics** toolbar, click **Edges** and choose **Ground**.

2 Select Edge 38 only.

3 In the **Settings** window for **Ground**, locate the **Shell Properties** section.

4 From the **Layered material** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

Electric Potential 1

1 In the **Physics** toolbar, click **Edges** and choose **Electric Potential**.

2 Select Edge 5 only.

3 In the **Settings** window for **Electric Potential**, locate the **Shell Properties** section.

4 From the **Layered material** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

5 Locate the **Electric Potential** section. In the V_0 text field, type `V_in`.

Continuity 1

1 In the **Physics** toolbar, click **Edges** and choose **Continuity**.

2 In the **Settings** window for **Continuity**, locate the **Layer Selection** section.

3 From the **Source** list, choose **Layered Material Stack 1 (stlmat1.zone2)**.

4 From the **Destination** list, choose **Layered Material Stack 1 (stlmat1.zone3)**.

5 In the **Selection** table, enter the following settings:

	Layered material	Offset (m)
√	Nichrome	0

MULTIPHYSICS

Electromagnetic Heating, Layered Shell 1 (ehls1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Boundary> Electromagnetic Heating, Layered Shell**.

GLOBAL DEFINITIONS

Silver Layer (mat2)

1. In the **Model Builder** window, under **Global Definitions>Materials** click **Silver Layer (mat2)**.
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	83e9	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	0.37	1	Young's modulus and Poisson's ratio
Density	rho	10500	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	420	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	230	J/(kg·K)	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	sigma_silver	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	1	1	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	18.9e-6	1/K	Basic

Nichrome Layer (mat3)

1. In the **Model Builder** window, click **Nichrome Layer (mat3)**.
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	`213e9`	Pa	Young's modulus and Poisson's ratio
Poisson's ratio	nu	`0.33`	1	Young's modulus and Poisson's ratio
Density	rho	`9000`	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`15`	W/(m·K)	Basic
Heat capacity at constant pressure	Cp	`20`	J/(kg·K)	Basic
Electrical conductivity	sigma_iso ; sigmaii = sigma_iso, sigmaij = 0	`sigma_nichrome`	S/m	Basic
Relative permittivity	epsilonr_iso ; epsilonrii = epsilonr_iso, epsilonrij = 0	`1`	1	Basic
Coefficient of thermal expansion	alpha_iso ; alphaii = alpha_iso, alphaij = 0	`10e-6`	1/K	Basic

MESH 1

Free Triangular 1

1 In the **Mesh** toolbar, click **More Generators** and choose **Free Triangular**.

2 In the **Settings** window for **Free Triangular**, locate the **Boundary Selection** section.

3 From the **Selection** list, choose **All boundaries**.

Size 1

1 Right-click **Free Triangular 1** and choose **Size**.

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

3 In the list, select **1**.

4 Click **Remove from Selection**.

5 Select Boundaries 2–4 only.

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

7 Locate the **Element Size Parameters** section.

8 Select the **Maximum element size** check box. In the associated text field, type `2`.

9 Click **Build All**.

STUDY 1

In the **Home** toolbar, click **Compute**.

RESULTS

Stress, Assembly (lshell)

In the **Settings** window for **3D Plot Group**, type `Stress, Assembly (lshell)` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Stress, Assembly (lshell)** node, then click **Surface 1**.

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

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

4 Click the **Scene Light** button in the **Graphics** toolbar.

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

6 In the **Stress, Assembly (lshell)** toolbar, click **Plot**.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.

2 Go to the **Add Predefined Plot** window.

3 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Stress, Slice (lshell)**.

4 Click **Add Plot** in the window toolbar.

RESULTS

Stress, Glass (lshell)

1 Drag and drop below **Stress, Assembly (lshell)**.

2 In the **Settings** window for **3D Plot Group**, type `Stress, Glass (lshell)` in the **Label** text field.

3 Click to expand the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Layered Material Slice: von Mises stress (MPa), Glass`.

Layered Material Slice 1

1 In the **Model Builder** window, expand the **Stress, Glass (lshell)** node, then click **Layered Material Slice 1**.

2 In the **Settings** window for **Layered Material Slice**, locate the **Expression** section.

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

4 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Physical**.

5 In the **Local z-coordinate** text field, type `0 d_glass`.

6 In the **Stress, Glass (lshell)** toolbar, click **Plot**.

Stress, Glass (lshell)

In the **Model Builder** window, right-click **Stress, Glass (lshell)** and choose **Duplicate**.

Stress, Conducting Layer (lshell)

1 In the **Model Builder** window, expand the **Results>Stress, Glass (lshell) 1** node, then click **Stress, Glass (lshell) 1**.

2 Drag and drop below **Stress, Glass (lshell)**.

3 In the **Settings** window for **3D Plot Group**, type `Stress, Conducting Layer (lshell)` in the **Label** text field.

4 Locate the **Title** section. In the **Title** text area, type `Layered Material Slice: von Mises stress (MPa), Conducting Layer`.

Layered Material Slice 1

1 In the **Model Builder** window, click **Layered Material Slice 1**.

2 In the **Settings** window for **Layered Material Slice**, locate the **Through-Thickness Location** section.

3 In the **Local z-coordinate** text field, type `d_glass+d_layer`.

4 In the **Stress, Conducting Layer (lshell)** toolbar, click **Plot**.

Temperature, Shell (htlsh)

1 In the **Model Builder** window, under **Results** click **Temperature, Shell (htlsh)**.

2 In the **Temperature, Shell (htlsh)** toolbar, click **Plot**.

Surface 1

1 In the **Model Builder** window, expand the **Results>Electric Potential (ecis)** node, then click **Surface 1**.

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

3 In the **Expression** text field, type `ecis.Vc`.

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

ADD PREDEFINED PLOT

1 Go to the **Add Predefined Plot** window.

2 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Geometry and Layup (lshell)>Shell Geometry (lshell)**.

3 Click **Add Plot** in the window toolbar.

4 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Shell Geometry (lshell)

1 Drag and drop below **Stress, Conducting Layer (lshell)**.

2 Right-click **Shell Geometry (lshell)** and choose **Duplicate**.

Stack Zones

1 In the **Model Builder** window, click **Shell Geometry (lshell) 1**.

2 Drag and drop below **Layer Cross-Section Preview**.

3 In the **Settings** window for **3D Plot Group**, type `Stack Zones` in the **Label** text field.

Surface 1

1 In the **Model Builder** window, expand the **Stack Zones** node, then click **Surface 1**.

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)>Layered materials>Layered Material Stack 1 (stlmat1)>stlmat1.zone - Zone index - 1**.

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

4 Click **Change Color Table**.

5 In the **Color Table** dialog box, select **Traffic>Traffic** in the tree.

6 Click **OK**.

7 In the **Stack Zones** toolbar, click **Plot**.

Surface Losses

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Surface Losses` in the **Label** text field.

Surface 1

1 In the **Surface Losses** 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)> Electric Currents in Layered Shells>Heating and losses>ecis.Qsh - Surface loss density, electromagnetic - W/m²**.

3 In the **Surface Losses** toolbar, click **Plot**.

Take the following steps to generate a plot of the norm of the surface traction vector in the surface plane.

Interface Stress

1 In the **Home** toolbar, click **Add Plot Group** and choose **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Interface Stress` in the **Label** text field.

3 Locate the **Title** section. From the **Title type** list, choose **Manual**.

4 In the **Title** text area, type `Layered Material Slice: Interface Stress (MPa)`.

Layered Material Slice 1

1 In the **Interface Stress** toolbar, click **More Plots** and choose **Layered Material Slice**.

2 In the **Settings** window for **Layered Material Slice**, locate the **Expression** section.

3 In the **Expression** text field, type `sqrt(lshell.sxz^2+lshell.syz^2)`.

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

5 Locate the **Through-Thickness Location** section. From the **Location definition** list, choose **Physical**.

6 In the **Local z-coordinate** text field, type `d_glass`.

Selection 1

1 Right-click **Layered Material Slice 1** and choose **Selection**.

2 Select Boundaries 2–4 only.

3 In the **Interface Stress** toolbar, click **Plot**.

Next, plot the glass plate’s deviation from being plane.

ADD PREDEFINED PLOT

1 In the **Home** toolbar, click **Add Predefined Plot** to open the **Add Predefined Plot** window.
2 Go to the **Add Predefined Plot** window.
3 In the tree, select **Study 1/Solution 1 (sol1)>Layered Shell>Warpage (wrp1)**.
4 Click **Add Plot** in the window toolbar.
5 In the **Home** toolbar, click **Add Predefined Plot** to close the **Add Predefined Plot** window.

RESULTS

Warping Displacement

1 In the **Model Builder** window, expand the **Warpage (wrp1)** node, then click **Warping Displacement**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Unit** field, type `um`.

Average Displacement

1 In the **Model Builder** window, click **Average Displacement**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 In the **Unit** field, type `um`.
4 In the **Warpage (wrp1)** toolbar, click **Plot**.

To calculate the values for the total generated heat and the integrated heat flux on the fluid side, perform a boundary integration. Before creating integration nodes, add a **Layered Material** dataset with evaluation set on interfaces.

Layered Material

1 In the **Model Builder** window, expand the **Results>Datasets** node.
2 Right-click **Results>Datasets>Layered Material** and choose **Duplicate**.

Layered Material 1

1 In the **Model Builder** window, click **Layered Material 1**.
2 In the **Settings** window for **Layered Material**, locate the **Layers** section.
3 From the **Evaluate in** list, choose **Interfaces**.

Surface Integration 1

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration>Surface Integration**.

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

3 From the **Dataset** list, choose **Layered Material 1**.

4 Locate the **Selection** section. From the **Selection** list, choose **All boundaries**.

5 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Heat Transfer in Shells>Boundary fluxes>htlsh.hfi2.q0 - Boundary convective heat flux - W/m²**.

6 Click **Evaluate**.

TABLE 1

1 Go to the **Table 1** window.

The result should be close to 8.5 W.

RESULTS

Surface Integration 2

1 In the **Results** toolbar, click 8.85 e-12 **More Derived Values** and choose **Integration>Surface Integration**.

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

3 From the **Dataset** list, choose **Layered Material 1**.

4 Select Boundaries 2–4 only.

5 Locate the **Through-Thickness Location** section. From the **Location input** list, choose **Manual**.

6 From the **Location definition** list, choose **Physical**.

7 Click **Replace Expression** in the upper-right corner of the **Expressions** section. From the menu, choose **Component 1 (comp1)>Electric Currents in Layered Shells>Heating and losses>ecis.Qsh - Surface loss density, electromagnetic - W/m²**.

8 Click **Evaluate**.

TABLE 2

1 Go to the **Table 2** window.

The result should be close to 13.8 W.