Model created in COMSOL Multiphysics 6.4

Heat Transfer in a Surface-Mount Package for a Silicon Chip

Introduction

All integrated circuits — especially high-speed devices — produce heat. In today's dense electronic system layouts heat sources are many times placed close to heat-sensitive ICs. Designers of printed-circuit boards often need to consider the relative placement of heat-sensitive and heat-producing devices, so that the sensitive ones do not overheat.

One type of heat-generating device is a voltage regulator, which can produce several watts of heat and reach a temperature higher than 70°C. If the board design places such a device close to a surface-mounted package that contains a sensitive silicon chip, the regulator's heat could cause reliability problems and failure due to overheating.

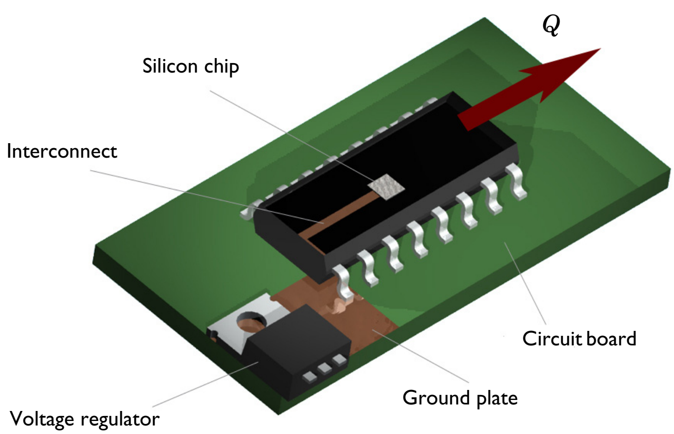


Figure 1: Layout of the simulated silicon device, its package, and a voltage regulator. The chip and the voltage regulator are connected through a ground plate, a pin, and the interconnect.

This simulation investigates the thermal situation for a silicon chip in a surface-mount package placed on a circuit board close to a hot voltage regulator. The chip is subjected to heat from the regulator and from internally generated heat.

Model Definition

The model is based on a SMD IC and voltage regulator layout as in Figure 1. The silicon chip sits in the center of the package and dissipates its heat to the surrounding environments. The chip also connects to a ground plane through an interconnect and one of the pins. A heat generating voltage regulator is placed on the same ground plane. This means that the voltage regulator may affect the silicon chip by the conducted heat and this may lead to overheating of the chip.

Heat transfers through the mount package to the surroundings through conduction according to:

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

The heat source, Q, is negligible in the circuit board, pins, and package, while in the chip, this model sets that parameter to a value equivalent to 20 mW. The conductivities of the components are chosen to be similar to:

- silicon, for the chip
- aluminum, for the pins
- FR4, for the PC board
- copper, for the ground plane and interconnect
- an arbitrary plastic, for the chip package

Heat dissipates from all air-exposed surfaces through forced heat convection, which is modeled using a heat transfer coefficient, h:

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\mathrm{inf}} - T)$$

The voltage regulator is simulated by setting a fixed temperature at that surface. The thin conducting layers of the ground plane and interconnect within the package is modeled using a 2D shell approximation, according to:

$$\nabla_{\mathbf{t}} \cdot (-d_{\mathrm{s}} k \nabla_{\mathbf{t}} T) = 0$$

where d_{s} is the layer's thickness, and $\nabla_{\mathbf{t}}$ represents the nabla operator projected onto the direction of the plane. The model uses a Heat Transfer interface to describe the 3D heat transfer as well as the 2D shell heat transfer.

Results and Discussions

Figure 2 illustrates the temperature distribution through the thickness. Being a good conductor, the interconnect delivers heat to the outer edge of the package, which gives the fairly constant temperature distribution around the interconnect.

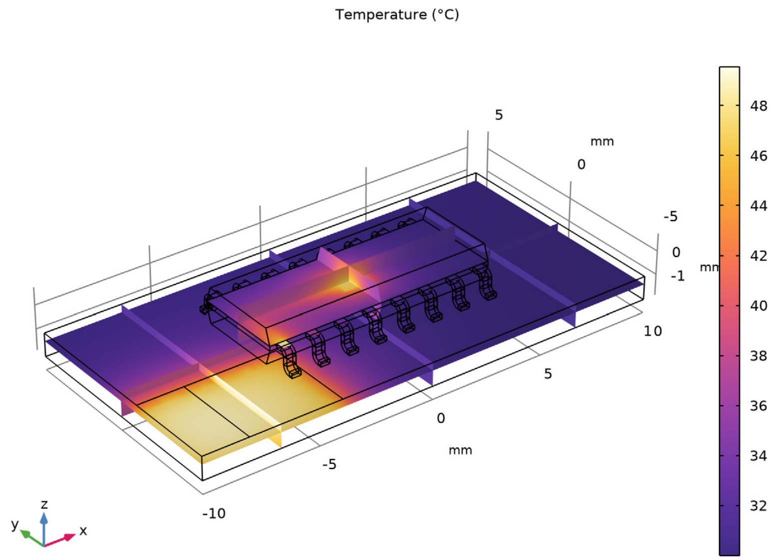


Figure 2: Slice plot of the temperature through the circuit board, interconnect, chip, and package. The effect of the interconnect is evident by its ability to conduct heat from the chip to the outer parts of the package.

An alternative view is achieved by using the transparency feature in the visualization tools of COMSOL Multiphysics. This results in a transparent 3D view of the temperature distribution, as shown in Figure 3. In that figure you can see the temperature distribution around the chip and along the interconnect.

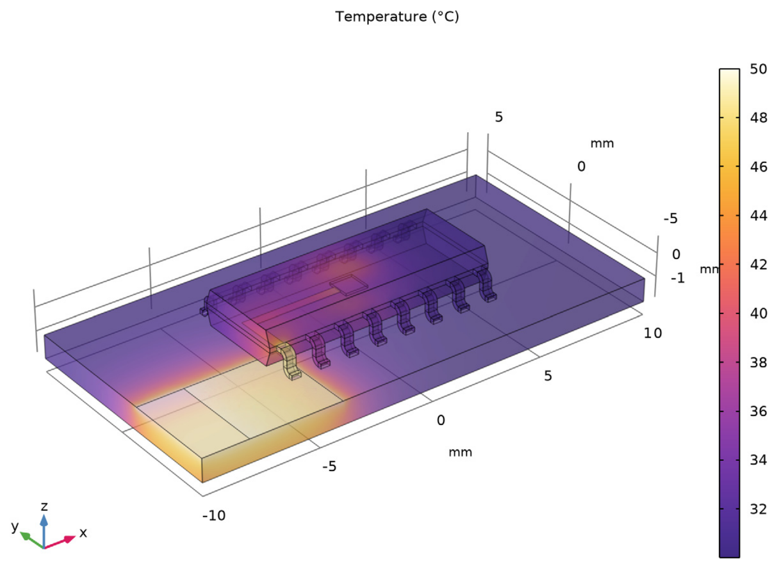


Figure 3: Boundary plot of the temperature created with the assistance of the transparency tool in COMSOL Multiphysics. This view also gives the temperature distribution on the chip and along the interconnect.

To get a closer look at the stationary temperature of the silicon chip, plot the temperature at the bottom boundary of the chip.

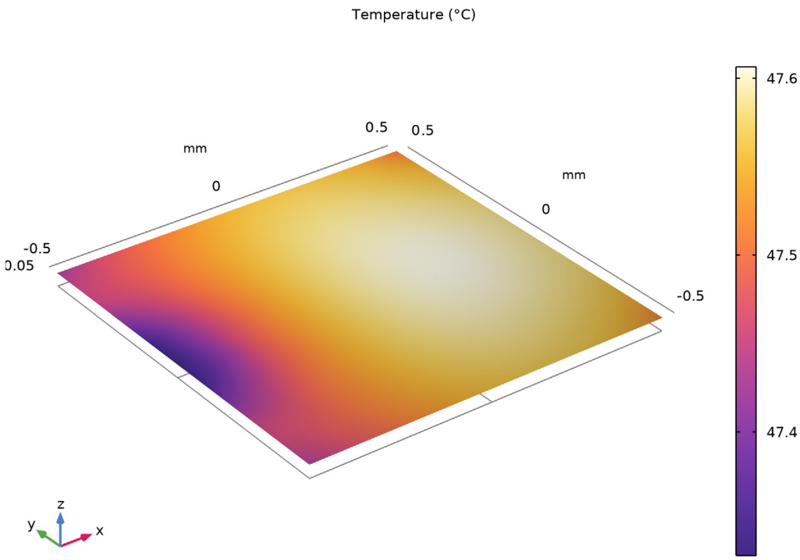


Figure 4: Temperature distribution on the bottom surface of the silicon chip.

The simulation predicts a maximum temperature of the silicon device of 47.6°C. This means that the device does not overheat in the present configuration.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/surface_mount_package`

Notes About the COMSOL Implementation

This tutorial uses the Heat Transfer interface from the Heat Transfer Module and, in particular, its Thin Layer feature. The thin layers of the modeled device are good heat conductor, so the Thin Layer feature is well-adapted for their representation. The two layers that have this definition are:

- The interconnect between the chip and the grounded pin.
- The ground plate that is also thermally connected to the temperature constraint coming from the voltage regulator.

While the numerical method considers these two modeling domains as interior boundaries, the model still includes a thickness to take the 3D heat flux into account.

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 **Heat Transfer** > **Heat Transfer in Solids (ht)**.

3 Click **Add**.

4 Click **Study**.

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

6 Click **Done**.

GEOMETRY 1

The geometry sequence for the model is available in a file. If you want to create it from scratch yourself, you can follow the instructions in the Geometry Modeling Instructions section. Otherwise, insert the geometry sequence as follows:

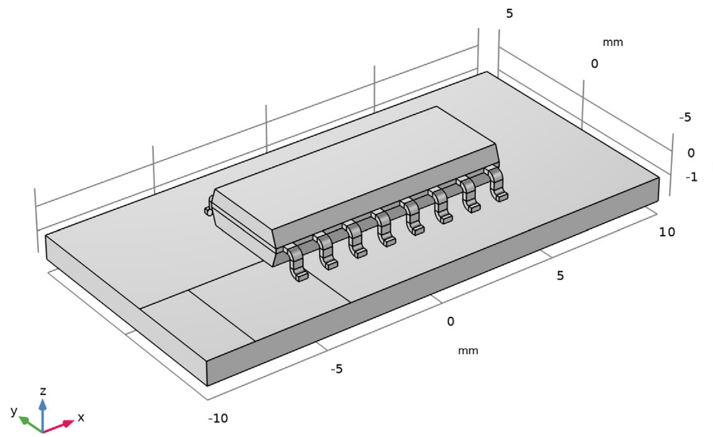
1 In the **Geometry** toolbar, click **Insert Sequence** and choose **Insert Sequence**.

2 Browse to the model's Application Libraries folder and double-click the file `surface_mount_package_geom_sequence.mph`.

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

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

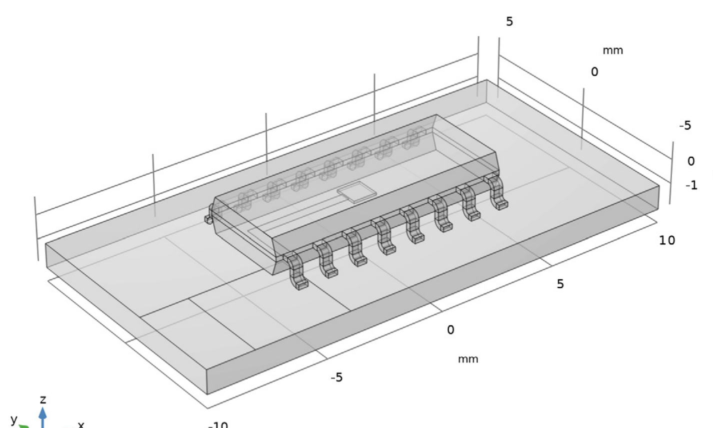
5 In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.



You should now see the geometry shown above.

The geometry includes the interconnect between the pin and the chip as well as the ground plate and the temperature surface resulting from the voltage regulator. To see the interconnect and the chip, you need to turn on transparency.

6 Click the **Transparency** button in the **Graphics** toolbar.



7 Click the Transparency button in the **Graphics** toolbar again to remove the transparency.

Now, define the domain settings including material properties, element order, heat source, and initial values.

MATERIALS

To define material properties for the model domains, use four predefined materials from the **Material Browser** and one custom material.

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** > **Aluminum**.
4. Click the **Add to Component** button in the window toolbar.
5. In the tree, select **Built-in** > **FR4 (Circuit Board)**.
6. Click the **Add to Component** button in the window toolbar.

MATERIALS

Aluminum (mat1)

1. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
2. From the **Selection** list, choose **Pins**.

FR4 (Circuit Board) (mat2)

1. In the **Model Builder** window, click **FR4 (Circuit Board) (mat2)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **PC Board**.

Plastic

1. In the **Materials** toolbar, click **Blank Material**.
2. In the **Settings** window for **Material**, type `Plastic` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Chip Package**.
4. In the **Model Builder** window, expand the **Component 1 (comp1)** > **Materials** > **Plastic (mat3)** node, then click **Basic (def)**.
5. In the **Settings** window for **Basic**, locate the **Output Properties** section.
6. Click **Select Quantity**.

7 In the **Physical Quantity** dialog, type `density` in the text field.

8 In the tree, select **General** > **Density (kg/m^3)**.

9 Click **OK**.

10 In the **Settings** window for **Basic**, locate the **Output Properties** section.

11 Click **Select Quantity**.

12 In the **Physical Quantity** dialog, type `heatcapacity` in the text field.

13 In the tree, select **Transport** > **Heat capacity at constant pressure (J/(kg*K))**.

14 Click **OK**.

15 In the **Settings** window for **Basic**, locate the **Output Properties** section.

16 Click **Select Quantity**.

17 In the **Physical Quantity** dialog, type `thermalconductivity` in the text field.

18 In the tree, select **Transport** > **Thermal conductivity (W/(m*K))**.

19 Click **OK**.

20 In the **Settings** window for **Basic**, locate the **Output Properties** section.

21 In the table, enter the following settings:

Property	Variable	Expression	Unit	Size
Density	rho	`2700[kg/m^3]`	kg/m³	1x1
Heat capacity at constant pressure	Cp	`900[J/(kg*K)]`	J/(kg·K)	1x1
Thermal conductivity	k_iso ; kii = k_iso, kij = 0	`0.2[W/(m*K)]`	W/(m·K)	3x3

ADD MATERIAL

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

2 In the tree, select **Built-in** > **Silicon**.

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

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

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

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

MATERIALS

Silicon (mat4)

1 In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.

2 From the **Selection** list, choose **Chip**.

Copper (mat5)

1 In the **Model Builder** window, click **Copper (mat5)**.

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

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

4 From the **Selection** list, choose **Copper Layers**.

This completes the materials settings.

HEAT TRANSFER IN SOLIDS (HT)

Heat Source 1

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.

2 Select Domain 11 only.

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

4 In the Q_0 text field, type `2e8[W/m^3]`.

This completes the domain settings. Now, set up the boundary conditions.

Heat Flux 1

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

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

3 From the **Selection** list, choose **Exterior Boundaries**.

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

5 In the h text field, type `50[W/(m^2*K)]`.

6 In the T_{ext} text field, type `30[degC]`.

Temperature 1

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

2 Select Boundary 4 only.

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

4 In the T_0 text field, type `50[degC]`.

Thin Layer 1

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

2 Select Boundary 7 only.

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

4 From the **Shell type** list, choose **Nonlayered shell**. Locate the **Layer Model** section. From the **Layer type** list, choose **Thermally thin approximation**.

Thin Layer 2

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

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

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

4 Locate the **Shell Properties** section. From the **Shell type** list, choose **Nonlayered shell**. In the L_{th} text field, type `5e-6[m]`.

5 Locate the **Layer Model** section. From the **Layer type** list, choose **Thermally thin approximation**.

MESH 1

Size 1

1 In the **Model Builder** window, under **Component 1 (comp1)** right-click **Mesh 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 4 and 7 only.

5 Locate the **Element Size** section. From the **Predefined** list, choose **Extra fine**.

Free Tetrahedral 1

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

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**.

The mesh should consist of around 16,500 elements.

STUDY 1

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

RESULTS

By default, you get a 3D plot of temperature. Note that the temperature is displayed in degrees Kelvin, which is the default temperature unit in the SI system.

To get a temperature plot in degrees Celsius, simply set the preferred unit for the temperature results.

Preferred Units 1

1 In the **Results** toolbar, click **Configurations** and choose **Preferred Units**.

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

3 Click **Add Physical Quantity**.

4 In the **Physical Quantity** dialog, select **General** > **Temperature (K)** in the tree.

5 Click **OK**.

6 In the **Settings** window for **Preferred Units**, locate the **Units** section.

7 In the table, enter the following settings:

Quantity	Unit	Preferred unit
Temperature	K	°C

8 Click **Apply**.

Nonlayered Shell

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Nonlayered Shell**.

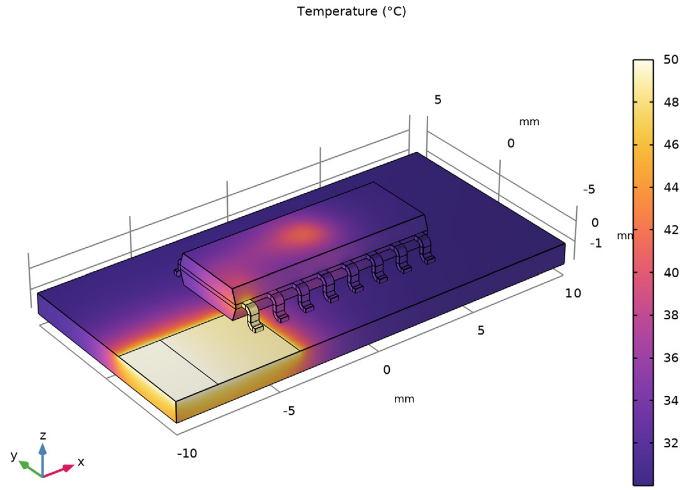
2 In the **Settings** window for **Surface Slit**, locate the **Expression on the Upside** section.

3 From the **Unit** list, choose **°C**.

4 Locate the **Expression on the Downside** section. From the **Unit** list, choose **°C**.

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

6 Click the **Go to Default View** button in the **Graphics** toolbar.



To see the chip as well, turn on transparency.

7 Click the **Transparency** button in the **Graphics** toolbar.

Compare the resulting plot to that in Figure 3.

8 Click the **Transparency** button in the **Graphics** toolbar again to remove the transparency.

RESULT TEMPLATES

1 In the **Results** toolbar, click **Result Templates** to open the **Result Templates** window.

2 Go to the **Result Templates** window.

Reproduce the temperature multislice plot in Figure 2 by adding a plot from the **Result Templates**.

3 In the tree, select **Study 1/Solution 1 (sol1)** > **Heat Transfer in Solids** > **Temperature, Multislice (ht)**.

4 Click the **Add Result Template** button in the window toolbar.

5 In the **Results** toolbar, click **Result Templates** to close the **Result Templates** window.

RESULTS

Multislice 1

1 In the **Model Builder** window, expand the **Temperature, Multislice (ht)** node, then click **Multislice 1**.
2 In the **Settings** window for **Multislice**, locate the **Multiplane Data** section.
3 Find the **X-planes** subsection. In the **Planes** text field, type `3`.
4 Find the **Z-planes** subsection. In the **Planes** text field, type `2`.
5 In the **Temperature, Multislice (ht)** toolbar, click **Plot**.
6 Click **Plot**.

Compare the result to that in Figure 2.

To visualize the temperature distribution on the silicon chip's bottom surface, follow the steps given below.

Temperature, Chip Surface

1 In the **Results** toolbar, click **3D Plot Group**.
2 In the **Settings** window for **3D Plot Group**, type `Temperature, Chip Surface` in the **Label** text field.
3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

Surface 1

1 In the **Temperature, Chip Surface** toolbar, click **Surface**.
2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.
3 From the **Color table** list, choose **HeatCameraLight**.

Selection 1

1 In the **Temperature, Chip Surface** toolbar, click **Selection**.
2 Select Boundary 195 only.
3 In the **Temperature, Chip Surface** toolbar, click **Plot**.
4 Click the **Zoom Extents** button in the **Graphics** toolbar.

Compare the resulting plot to that in Figure 4.

Geometry Modeling Instructions

If you want to create the geometry yourself, follow these steps.

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**.

PC Board

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

2 In the **Settings** window for **Block**, type `PC Board` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Width** text field, type `20`.

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

5 Locate the **Position** section. In the **x** text field, type `-10`.

6 In the **y** text field, type `-5`.

7 In the **z** text field, type `-1.9`.

8 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

Block 2 (blk2)

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

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

3 In the **Width** text field, type `9.9`.

4 In the **Depth** text field, type `3.9`.

5 In the **Height** text field, type `0.2`.

6 Locate the **Position** section. From the **Base** list, choose **Center**.

Hexahedron 1 (hex1)

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

2 In the **Settings** window for **Hexahedron**, locate the **Vertices** section.

3 In row **1**, set **x** to `-4.95`.

4 In row **2**, set **x** to `4.95`.

5 In row **3**, set **x** to `4.95`.

6 In row **4**, set **x** to `-4.95`.

7 In row **5**, set **x** to `-4.95`.

8 In row **6**, set **x** to `4.95`.

9 In row **7**, set **x** to `4.95`.

10 In row **8**, set **x** to `-4.95`.

11 In row **1**, set **y** to `-1.95`.

12 In row **2**, set **y** to `-1.95`.

13 In row **3**, set **y** to `1.95`.

14 In row **4**, set **y** to `1.95`.

15 In row **5**, set **y** to `-1.713419348`.

16 In row **6**, set **y** to `-1.713419348`.

17 In row **7**, set **y** to `1.713419348`.

18 In row **8**, set **y** to `1.713419348`.

19 In row **1**, set **z** to `0.1`.

20 In row **2**, set **z** to `0.1`.

21 In row **3**, set **z** to `0.1`.

22 In row **4**, set **z** to `0.1`.

23 In row **5**, set **z** to `0.75`.

24 In row **6**, set **z** to `0.75`.

25 In row **7**, set **z** to `0.75`.

26 In row **8**, set **z** to `0.75`.

27 In row **5**, set **y** to `-1.72`.

28 In row **6**, set **y** to `-1.72`.

29 In row **7**, set **y** to `1.72`.

30 In row **8**, set **y** to `1.72`.

Mirror 1 (mir1)

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

2 Select the object **hex1** only.

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

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

Chip Package

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

2 In the **Settings** window for **Union**, type `Chip Package` in the **Label** text field.

3 Select the objects **blk2**, **hex1**, and **mir1** only.

4 Locate the **Union** section. Clear the **Keep interior boundaries** checkbox.

5 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

Block 3 (blk3)

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

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

3 In the **Width** text field, type `0.4`.

4 In the **Depth** text field, type `0.26`.

5 In the **Height** text field, type `0.2`.

6 Locate the **Position** section. In the **x** text field, type `-4.645`.

7 In the **y** text field, type `-2.21`.

8 In the **z** text field, type `-0.1`.

Revolve 1 (rev1)

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

2 On the object **blk3**, select Boundary 3 only.

3 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

4 Click the **Angles** button.

5 In the **End angle** text field, type `90`.

6 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.

7 Find the **Point on the revolution axis** subsection. In the **y** text field, type `-2.211`.

8 In the **z** text field, type `-0.24`.

9 Find the **Direction of revolution axis** subsection. In the **x** text field, type `1`.

10 In the **y** text field, type `0`.

Extrude 1 (ext1)

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

2 On the object **rev1**, select Boundary 2 only.

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

4 In the table, enter the following settings:

Distances (mm)
0.322

Revolve 2 (rev2)

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

2 On the object **ext1**, select Boundary 3 only.

3 In the **Settings** window for **Revolve**, locate the **Revolution Angles** section.

4 Click the **Angles** button.

5 In the **End angle** text field, type `-90`.

6 Locate the **Revolution Axis** section. From the **Axis type** list, choose **3D**.

7 Find the **Point on the revolution axis** subsection. In the **y** text field, type `-2.69`.

8 In the **z** text field, type `-0.561`.

9 Find the **Direction of revolution axis** subsection. In the **x** text field, type `1`.

10 In the **y** text field, type `0`.

Extrude 2 (ext2)

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

2 On the object **rev2**, select Boundary 2 only.

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

4 In the table, enter the following settings:

Distances (mm)
0.16

Union 2 (uni2)

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

2 Select the object **ext2** only.

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

4 Clear the **Keep interior boundaries** checkbox.

Array 1 (arr1)

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

2 Select the object **uni2** only.

3 In the **Settings** window for **Array**, locate the **Size** section.

4 In the **x size** text field, type `8`.

5 Locate the **Displacement** section. In the **x** text field, type `1.27`.

6 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. Click **New**.

7 In the **New Cumulative Selection** dialog, type `Pins` in the **Name** text field.

8 Click **OK**.

Mirror 2 (mir2)

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

2 In the **Settings** window for **Mirror**, locate the **Input** section.

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

4 From the **Input objects** list, choose **Pins**.

5 Locate the **Selections of Resulting Entities** section. Find the **Cumulative selection** subsection. From the **Contribute to** list, choose **Pins**.

6 Locate the **Normal Vector to Plane of Reflection** section. In the **y** text field, type `1`.

7 In the **z** text field, type `0`.

Work Plane 1 (wp1)

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

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

3 In the **z-coordinate** text field, type `-0.9`.

Work Plane 1 (wp1) > Plane Geometry

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

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 `6`.

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

5 Locate the **Position** section. In the **xw** text field, type `-10`.

6 In the **yw** text field, type `-5`.

7 Click to expand the **Layers** section. In the table, enter the following settings:

Layer name	Thickness (mm)
`Layer 1`	`2`

8 Clear the **Layers on bottom** checkbox.

9 Select the **Layers to the left** checkbox.

Interconnect

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Work Plane 1 (wp1)** and choose **Work Plane**.

2 In the **Settings** window for **Work Plane**, type `Interconnect` in the **Label** text field.

3 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

Interconnect (wp2) > Plane Geometry

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

Interconnect (wp2) > 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 `4.145`.

4 In the **Height** text field, type `2.15`.

5 Locate the **Position** section. In the **xw** text field, type `-4.645`.

6 In the **yw** text field, type `-1.95`.

Interconnect (wp2) > Rectangle 2 (r2)

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 `3.745`.

4 In the **Height** text field, type `1.75`.

5 Locate the **Position** section. In the **xw** text field, type `-4.245`.

6 In the **yw** text field, type `-1.95`.

Interconnect (wp2) > Difference 1 (dif1)

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

2 Select the object **r1** 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 **r2** only.

Chip

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** right-click **Interconnect (wp2)** and choose **Block**.

2 In the **Settings** window for **Block**, type `Chip` in the **Label** text field.

3 Locate the **Size and Shape** section. In the **Height** text field, type `0.1`.

4 Locate the **Selections of Resulting Entities** section. Select the **Resulting objects selection** checkbox.

5 Locate the **Position** section. From the **Base** list, choose **Center**.

Form Union (fin)

1 In the **Model Builder** window, click **Form Union (fin)**.

2 In the **Settings** window for **Form Union/Assembly**, click **Build Selected**.

Geometry

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, type `Geometry` in the **Label** text field.

3 Locate the **Entities to Select** section. From the **Geometric entity level** list, choose **Object**.

4 Select the object **fin** only.

Exterior Boundaries

1 In the **Geometry** toolbar, click **Selections** and choose **Adjacent Selection**.

2 In the **Settings** window for **Adjacent Selection**, type `Exterior Boundaries` in the **Label** text field.

3 Locate the **Input Entities** section. Click **Add**.

4 In the **Add** dialog, select **Geometry** in the **Input selections** list.

5 Click **OK**.

Copper Layers

1 In the **Geometry** toolbar, click **Selections** and choose **Explicit Selection**.

2 In the **Settings** window for **Explicit Selection**, locate the **Entities to Select** section.

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

4 On the object **fin**, select Boundaries 7 and 37 only.

5 In the **Label** text field, type `Copper Layers`.