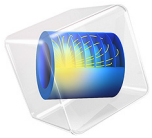


Thermal Analysis of a High-Power IGBT Module

Introduction

The insulated-gate bipolar transistor module (IGBT module) is a popular choice in high-power systems thanks to its capability to resist high voltages and to channel strong currents while switching between the two modes rapidly. This industrial-scale proof-of-concept (POC) model demonstrates how to perform an electric–thermal analysis of a high-power IGBT module. The module has a rated voltage of 1200 V and operates at a nominal current of 1800 A.

The module consists of a number of IGBT dies mounted on top of a copper base plate with a heat sink at the bottom. In the module electric currents generate heat due to resistive loss, which is also known as *Joule heating*. While the heat sink expels the heat at a relatively constant rate, the switching of the module and subsequent increase and decrease of the current density and the heat source causes the module to heat up and cool down in a cyclic fashion. This repeated thermal expansion and mechanical deformation then leads to mechanical fatigue[1], in particular in the attachment points between the bonding wires and the die metalization layer.

In this model the current density distribution, the heat source, and the temperature distribution are investigated under stationary conditions to get a better understanding of this phenomenon and how it is affected by the module's design.

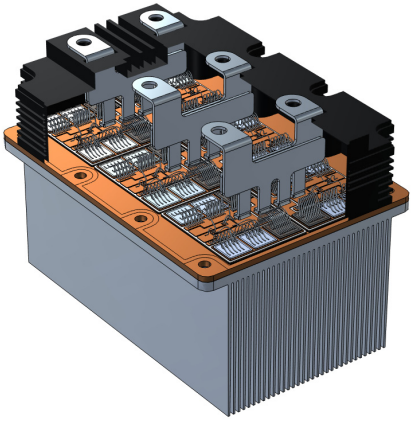


Figure 1: An overview of the IGBT module with parts of the plastic cover removed for visualization purposes. Six terminals are available at the top of the device, at which the current throughput is specified. At the terminals, a fixed temperature is used. At the bottom of the device a heat sink is used with convective heat flux boundary conditions.

1. Note that other effects include thermal ratcheting, creep, delamination, cracking, electromigration.

Model Definition

Figure 1 shows an overview of the IGBT module. The module is divided into six sections, with each section organized as in Figure 2. During the numerical analysis, the electric potential and temperature are investigated using the *Electric Currents* and the *Heat Transfer in Solids* physics interfaces, respectively. To couple the two physics, the *Electromagnetic Heating* multiphysics coupling is used.

The electric properties of the semiconductor material have been included as a macroscopic effect: an *effective conductivity* is applied such that the potential drop across the junction agrees with a predetermined (temperature-dependent) value as given by Ref. 1.

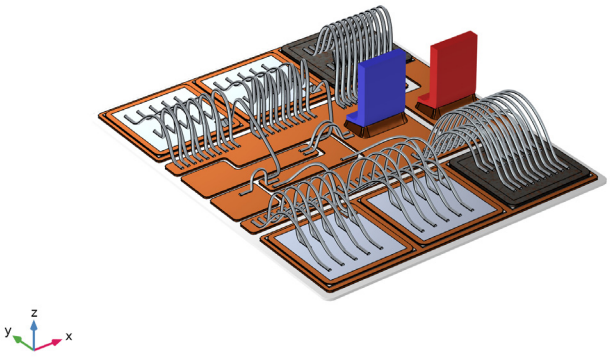


Figure 2: One of six sections of the module. Each section consists of four IGBT chips (silver-colored) and two FWDs (black), connected via aluminum bond wires and copper base plates. The current input and output are marked as red and blue in the figure.

Using a stationary study, the device is modeled in the "on" state, where strong currents pass through the IGBT dies and a relatively small portion of the current is leaking back through the freewheeling diodes (FWDs). The generated heat is dissipated through a heat sink, and both the currents and the temperature profile are evaluated in the semiconductors, the metalization layers, and the bond wires.

The equations for the electric currents are only solved in the electric conductors and the semiconductors, while the heat equation is also solved in the aluminum oxide layer and the heat sink. The heat transfer through the terminals is modeled using a fixed temperature boundary condition and the heat transfer from the heat sink to the air is modeled using a convective heat flux boundary condition:

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\text{ext}} - T)\,. \tag{1}$$

Here, h is the heat transfer coefficient, $T_{\text{ext}} = 60°\text{C}$ is the air temperature and $\mathbf{n} \cdot \mathbf{q}$ is the outward heat flux. The remaining boundaries are assumed to be thermally insulating.

For the semiconductor conductivity, the model uses a measured relation between the temperature of the IGBT die and its collector–emitter voltage in the module's on state, as given by Ref. 1:

$$V_{\text{ce}} = -0.00235T + 0.544 \tag{2}$$

where V_{ce} is given in volts and T in degrees Celsius. This is implemented using an auxiliary degree of freedom to find the conductivity required to get the measured collector-emitter voltage.

Results and Discussion

The electric potential of one section of the module is shown in Figure 3. The figure illustrates how the majority of the voltage drop over the current path occurs across the transistor junctions. The corresponding current density distribution is shown in Figure 4, which shows that only a small portion of the current leaks through the FWDs. The highest current densities are reached at the connection points between the metalization layers of the IGBT dies and the bond wires.

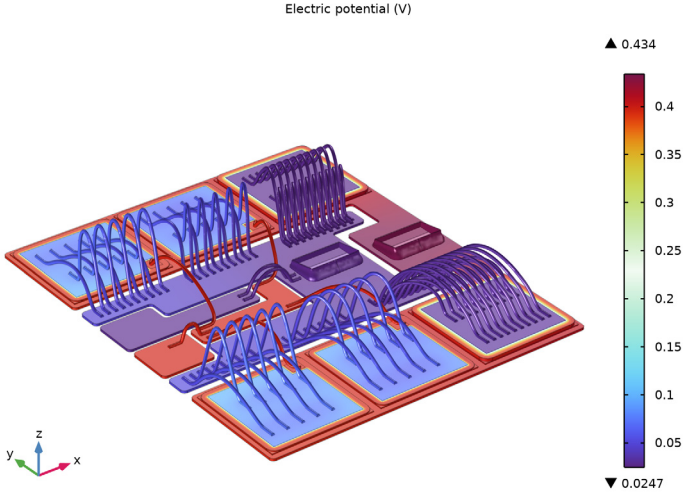


Figure 3: Electric potential distribution of one section. The current path goes through the IGBT chips, introducing a voltage drop which is in agreement with the measured value.

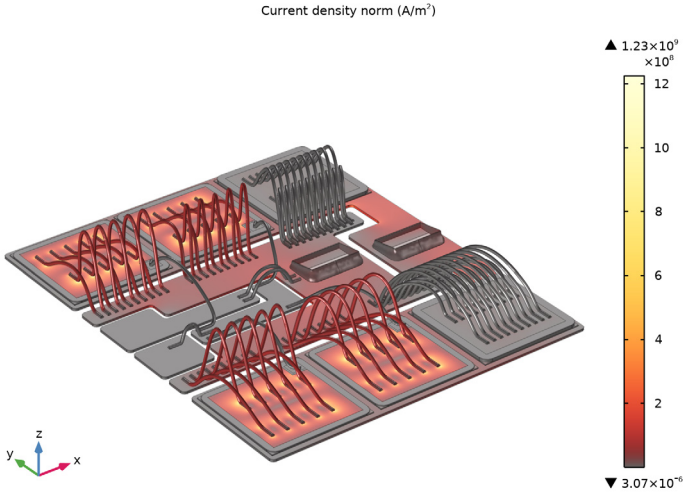


Figure 4: Current density distribution of one section. The highest current density is reached at the base of the bond wires. Only a small portion of the current leaks through the FWDs.

The temperature distribution over the same section is shown in Figure 5. The maximum temperature in the section is approximately 110°C, 50 degrees higher than the external air temperature as defined on the heat sinks and 40 degrees higher than the temperature as defined on the busbar terminal boundaries. The temperature distribution of the entire system is illustrated in Figure 6.

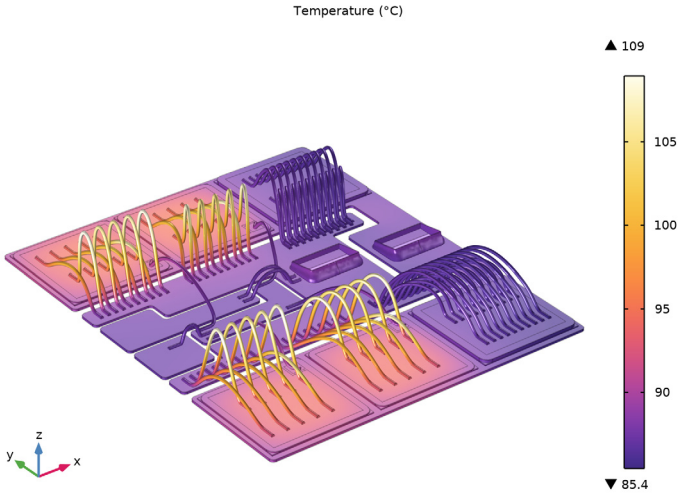


Figure 5: Temperature distribution of one segment in the IGBT module. The maximum temperature is approximately 110°C, and is reached in the wires connecting the IGBT dies with the copper plates.

Table 1 shows some key statistics of the IGBT dies. The average collector–emitter voltage is 324 mV, which makes up the majority of the total voltage drop over the current path. The average junction temperature is 93.5°C.

TABLE 1: RESULTS FROM THE IGBT MODULE IN THE ON STATE.

Description	Value
Average current density across junction (A/cm^2)	52.6
Average junction temperature (°C)	93.5
Collector-emitter voltage (mV)	324
Effective IGBT conductivity (S/m)	227

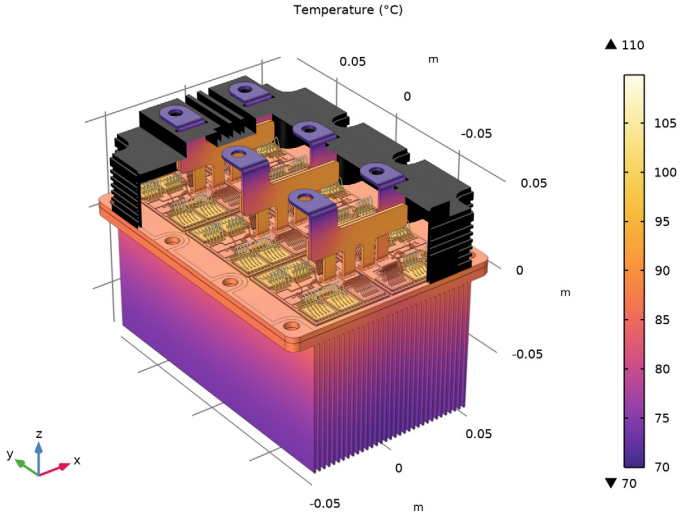


Figure 6: Temperature distribution of the entire IGBT module. The highest temperature was reached in the bond wires in the middle sections.

Reference

1. A. Tong and others, "Comparative Study of the Parameter Acquisition Methods for the Cauer Thermal Network Model of an IGBT Module," *Electronics*, vol. 12, no. 7, art. 1650, 2023.

Article available at: www.mdpi.com/2079-9292/12/7/1650.

The *Thermal Analysis of a High-Power IGBT Module* tutorial model uses several dimensions, material properties, and the expression for the collector-emitter voltage as given by the article (in particular, as given by *Figure 4*, *Table 2*, and *Table 4*).

Application Library path: `ACDC_Module/Electromagnetic_Heating/igbt_joule_heating`

At this point, there are two options. One is to go to section Appendix in this tutorial and follow the instructions for building the geometry, selections and mesh there. In case you have limited interest in doing that yourself, start by opening `igbt_joule_heating_introduction.mph`.

1 From the **File** menu, choose **Open**.
2 Browse to the model's Application Libraries folder and double-click the file `igbt_joule_heating_introduction.mph`.
3 From the **File** menu, choose **Save As**.
4 Browse to a suitable folder and type the filename `igbt_joule_heating.mph`.

Now, proceed with the setup of the physics interfaces. This model is centered around the *Electric Currents* and *Heat Transfer in Solids* interfaces. The two interfaces will be connected using the *Electromagnetic Heating* multiphysics coupling, feeding the electromagnetic loss as a heat source into the thermal analysis.

ADD PHYSICS

1 In the **Home** toolbar, click **Add Physics** to open the **Add Physics** window.
2 Go to the **Add Physics** window.
3 In the tree, select **AC/DC** > **Electric Fields and Currents** > **Electric Currents (ec)**.
4 Click the **Add to Component 1** button in the window toolbar.

ELECTRIC CURRENTS (EC)

1 In the **Settings** window for **Electric Currents**, locate the **Domain Selection** section.
2 From the **Selection** list, choose **Electric Current Domains**.
3 Click to expand the **Discretization** section. From the **Electric potential** list, choose **Linear**.

Ground 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Ground**.
2 In the **Settings** window for **Ground**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Ground (Import 1)**.

Boundary Terminal 1

1 In the **Physics** toolbar, click **Boundaries** and choose **Boundary Terminal**.
2 In the **Settings** window for **Boundary Terminal**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Terminal (Import 1)**.

4 Locate the **Terminal** section. In the I_0 text field, type `I_col`.

Electric Shielding 1

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

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

3 From the **Selection** list, choose **Aluminum (Metalization)**.

4 Locate the **Thickness** section. In the d_s text field, type `th_met`.

ADD PHYSICS

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

2 In the tree, select **Heat Transfer** > **Heat Transfer in Solids (ht)**.

3 Click the **Add to Component 1** button in the window toolbar.

HEAT TRANSFER IN SOLIDS (HT)

1 In the **Settings** window for **Heat Transfer in Solids**, locate the **Domain Selection** section.

2 From the **Selection** list, choose **Heat Transfer Domains**.

3 Click to expand the **Discretization** section. From the **Temperature** list, choose **Linear**.

Initial Values 1

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

2 In the **Settings** window for **Initial Values**, locate the **Initial Values** section.

3 In the T text field, type `T_trm`.

Temperature 1

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

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

3 From the **Selection** list, choose **Fixed Temperature**.

4 Locate the **Temperature** section. In the T_0 text field, type `T_trm`.

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 **Forced Convection (Import 1)**.

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

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

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

Thin Layer 1

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 **Aluminum (Metalization)**.

4 Locate the **Shell Properties** section. From the **Shell type** list, choose **Nonlayered shell**. In the L_{th} text field, type `th_met`.

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

MULTIPHYSICS

Electromagnetic Heating 1 (emh1)

In the **Physics** toolbar, click **Multiphysics Couplings** and choose **Domain** > **Electromagnetic Heating**.

Now, you have finished setting up the physics. What remains before adding the study is to add an auxiliary degree of freedom to allow COMSOL to determine the semiconductor conductivity required to match the measured temperature dependent collector-emitter voltage as given by Equation 2. Start by including the measured curve in the model.

GLOBAL DEFINITIONS

Collector-Emitter Voltage IGBT (Open)

1 In the **Home** toolbar, click **Functions** and choose **Global** > **Analytic**.

2 In the **Settings** window for **Analytic**, type `Collector-Emitter Voltage IGBT (Open)` in the **Label** text field.

3 In the **Function name** text field, type `V_col`.

4 Locate the **Definition** section. In the **Expression** text field, type `-0.00235[V/degC]*T+0.544[V]`.

5 In the **Arguments** text field, type `T`.

6 Locate the **Units** section. In the **Function** text field, type `V`.

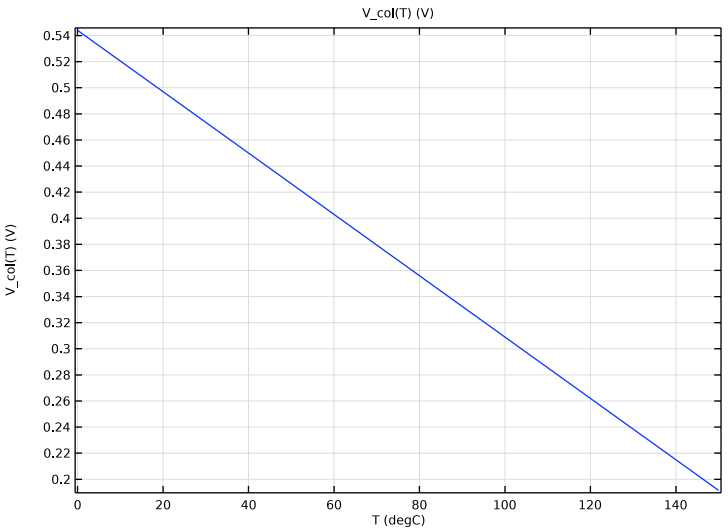
7 In the table, enter the following settings:

Argument	Unit
T	degC

8 Locate the **Plot Parameters** section. In the table, enter the following settings:

Plot	Argument	Lower limit	Upper limit	Fixed value	Unit
√	T	0[degC]	150[degC]	0	K

9 Click **Plot**.



DEFINITIONS

Add a nonlocal coupling to determine the average current density and the average temperature at the IGBT junction.

Domain Average IGBT

1 In the **Definitions** toolbar, click **Nonlocal Couplings** and choose **Average**.

2 In the **Settings** window for **Average**, type `Domain Average IGBT` in the **Label** text field.

3 In the **Operator name** text field, type `av_igbt`.

4 Locate the **Source Selection** section. From the **Selection** list, choose **IGBT Integration Domain**.

Now, add a global equation to add a degree of freedom for the average current density in the IGBT.

ADD PHYSICS

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

2 In the tree, select **Mathematics** > **ODE and DAE Interfaces** > **Global ODEs and DAEs (ge)**.

3 Click the **Add to Component 1** button in the window toolbar.

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

GLOBAL ODES AND DAES (GE)

Current Density IGBT

1 In the **Settings** window for **Global Equations**, type `Current Density IGBT` in the **Label** text field.

2 Locate the **Global Equations** section. In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
J_igbt	J_igbt-av_igbt(ec.normJ)	I_col/(24*(13.5[mm])^2)	0	Current density across junction (average)

3 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

4 In the **Physical Quantity** dialog, select **Electromagnetics** > **Current density (A/m^2)** in the tree.

5 Click **OK**.

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

7 Click **Select Source Term Quantity**.

8 In the **Physical Quantity** dialog, select **Electromagnetics** > **Current density (A/m^2)** in the tree.

9 Click **OK**.

Temperature IGBT

1 In the **Global ODEs and DAEs** toolbar, click **Global Equations**.

2 In the **Settings** window for **Global Equations**, type `Temperature IGBT` in the **Label** text field.

3 Locate the **Global Equations** section. In the table, enter the following settings:

Name	f(u,ut,utt,t) (1)	Initial value (u_0) (1)	Initial value (ut_0) (1/s)	Description
T_igbt	T_igbt-av_igbt(T)	T_trm	0	Junction temperature (average)

4 Locate the **Units** section. Click **Select Dependent Variable Quantity**.

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

6 Click **OK**.

7 In the **Settings** window for **Global Equations**, locate the **Units** section.

8 Click **Select Source Term Quantity**.

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

10 Click **OK**.

DEFINITIONS

Define the collector-emitter voltage at the average junction temperature as well as the effective conductivity.

Effective Conductivity IGBT

1 In the **Definitions** toolbar, click **Local Variables**.

2 In the **Settings** window for **Variables**, type `Effective Conductivity IGBT` in the **Label** text field.

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

Name	Expression	Unit	Description
`V_igbt`	`V_col(T_igbt)`	V	`Collector-emitter voltage IGBT (open, approximate)`
`s_igbt`	`J_igbt/(V_igbt/th_igbt)`	S/m	`Effective conductivity IGBT (open, approximate)`

MATERIALS

Now, you will see that COMSOL starts detecting missing material properties. The properties that should be added are listed in the following table. Please check all of them for the correct value, even the ones that are already filled in.

Furthermore, note that material properties marked with an "`x`" are not used at all (they may be present, but their value is irrelevant). Common practice is to include the unit when typing: "`57[W/(m*K)]`".

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Materials**, add the following material properties:

	sigma [S/m]	epsr	k [W/(m*K)]	rho [kg/m^3]	Cp [J/(kg*K)]
mat1	1/(1.04e-1[mohm*mm])	1	57	7.30e3	230
mat2	1/(1.68e-2[mohm*mm])	1	400	8.92e3	380
mat3	1/(2.65e-2[mohm*mm])	1	237	2.70e3	900
mat4	10	1	148	2.33e3	700
mat5	s_igbt	1	148	2.33e3	700
mat6	1/(2.65e-2[mohm*mm])	1	237	2.70e3	900
mat7	x	x	20	3.96e3	753
mat8	x	x	3.5	3.7[g/cm^3]	1.2[J/(g*K)]

Hide for Geometry 1

1 In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** > **View 1** node.

2 Right-click **Hide for Geometry 1** and choose **Disable**.

ADD STUDY

With the physics and the materials ready, it is now time to add a study. The module will be analyzed in its 'on' state, which corresponds to a nominal current of 1800 A and a rated voltage of 1200 V. The electric potential and temperature distribution will be computed and global equations are included to find the effective conductivity of the IGBT chips. The study will assume stationary conditions.

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

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

3 Find the **Studies** subsection. In the **Select Study** tree, select **General Studies** > **Stationary**.

4 Click the **Add Study** button in the window toolbar.

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

Now, edit the default solver to improve convergence speed and robustness.

STUDY 1

Solution 1 (sol1)

1 In the **Study** toolbar, click **Show Default Solver**.

2 In the **Model Builder** window, expand the **Solution 1 (sol1)** node.

3 In the **Model Builder** window, expand the **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **Segregated 1** node, then click **Electric Currents**.

4 In the **Settings** window for **Segregated Step**, click to expand the **Method and Termination** section.

5 In the **Damping factor** text field, type 1.

6 In the **Model Builder** window, under **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **Segregated 1** click **Temperature**.

7 In the **Settings** window for **Segregated Step**, locate the **Method and Termination** section.

8 In the **Damping factor** text field, type 1.

9 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **Segregated 1** > **Temperature** and choose **Move Up**.

10 Right-click **Study 1** > **Solver Configurations** > **Solution 1 (sol1)** > **Stationary Solver 1** > **Segregated 1** > **Lower Limit 1** and choose **Move Up**.

11 In the **Model Builder** window, click **Study 1**.

12 In the **Settings** window for **Study**, locate the **Study Settings** section.

13 Clear the **Generate default plots** checkbox.

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

RESULTS

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

2 In the **Settings** window for **Results**, locate the **Update of Results** section.

3 Select the **Only plot when requested** checkbox, since the resulting plots will take a while to evaluate.

The results will be divided into three node groups, one for a single set of dies, one for a section of the module, and one for the entire model. Create and modify three datasets corresponding to these groups.

4 In the **Model Builder** window, expand the **Results** node.

Study 1/Solution 1 (2) (sol1)

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

2 Right-click **Results** > **Datasets** > **Study 1/Solution 1 (sol1)** and choose **Duplicate**.

Study 1/Solution 1 (3) (sol1)

In the **Model Builder** window, right-click **Study 1/Solution 1 (2) (sol1)** and choose **Duplicate**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Set of Dies (Domain)**.

Study 1/Solution 1 (2) (sol1)

In the **Model Builder** window, under **Results** > **Datasets** click **Study 1/Solution 1 (2) (sol1)**.

Selection

1 In the **Results** toolbar, click **Attributes** and choose **Selection**.

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

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

4 From the **Selection** list, choose **Module Section (Domain)**.

Electric Potential (ec)

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Electric Potential (ec)` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

5 Click to expand the **Quality** section. From the **Smoothing** list, choose **Inside geometry domains**.

Surface 1

1 Right-click **Electric Potential (ec)** and choose **Surface**.

2 In the **Settings** window for **Surface**, locate the **Coloring and Style** section.

3 From the **Color table** list, choose **Dipole**.

4 From the **Color table transformation** list, choose **Nonlinear symmetric**.

5 In the **Color calibration parameter** text field, type `0.8`.

Line 1

1 In the **Model Builder** window, right-click **Electric Potential (ec)** and choose **Line**.

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

3 In the **Expression** text field, type `0`.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Black**.

7 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

Transparency 1

1 Right-click **Line 1** and choose **Transparency**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.75`.

4 Click the **Show More Options** button in the **Model Builder** toolbar.

5 In the **Show More Options** dialog, select **Results** > **Views** in the tree.

6 In the tree, select the checkbox for the node **Results** > **Views**.

7 Click **OK**.

View 3D 2

In the **Model Builder** window, under **Results** right-click **Views** and choose **View 3D**.

Camera

1 Click the **Show Grid** button in the **Graphics** toolbar.

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

3 In the **Model Builder** window, expand the **View 3D 2** node, then click **Camera**.

4 In the **Settings** window for **Camera**, click **Update**.

Electric Potential (ec)

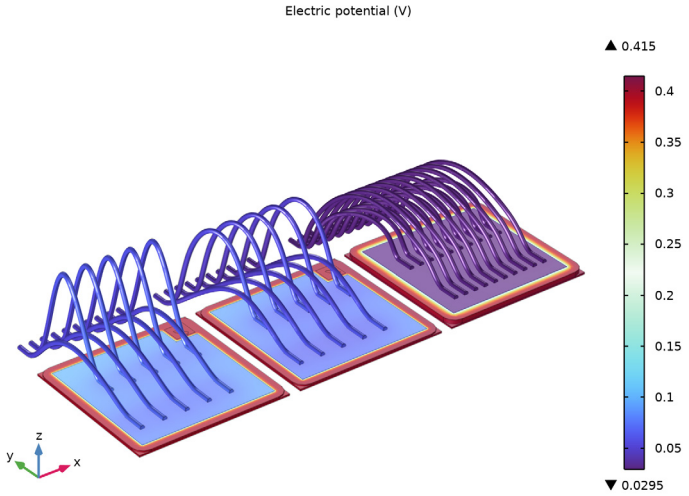
1 In the **Model Builder** window, under **Results** click **Electric Potential (ec)**.

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

3 From the **View** list, choose **View 3D 2**.

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

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



Current Density Norm (ec)

1 In the **Results** toolbar, click **3D Plot Group**.

2 In the **Settings** window for **3D Plot Group**, type `Current Density Norm (ec)` in the **Label** text field.

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

5 Locate the **Quality** section. From the **Smoothing** list, choose **Inside geometry domains**.

Surface 1

1 Right-click **Current Density Norm (ec)** and choose **Surface**.

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

3 In the **Expression** text field, type `ec.normJ`.

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

5 From the **Color table transformation** list, choose **Nonlinear**.

6 In the **Color calibration parameter** text field, type `-0.8`.

7 Click to expand the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

Selection 1

1 Right-click **Surface 1** and choose **Selection**.

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

3 From the **Selection** list, choose **Set of Dies (Not Metalization)**.

Surface 2

1 In the **Model Builder** window, right-click **Current Density Norm (ec)** and choose **Surface**.

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

3 In the **Expression** text field, type `ec.normJs/th_met`.

4 Click to expand the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

6 Click to expand the **Inherit Style** section. From the **Plot** list, choose **Surface 1**.

Selection 1

1 Right-click **Surface 2** and choose **Selection**.

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

3 From the **Selection** list, choose **Set of Dies (Metalization)**.

Line 1

1 In the **Model Builder** window, right-click **Current Density Norm (ec)** and choose **Line**.

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

3 In the **Expression** text field, type `0`.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Black**.

7 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

Transparency 1

1 Right-click **Line 1** and choose **Transparency**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.75`.

Current Density Norm (ec)

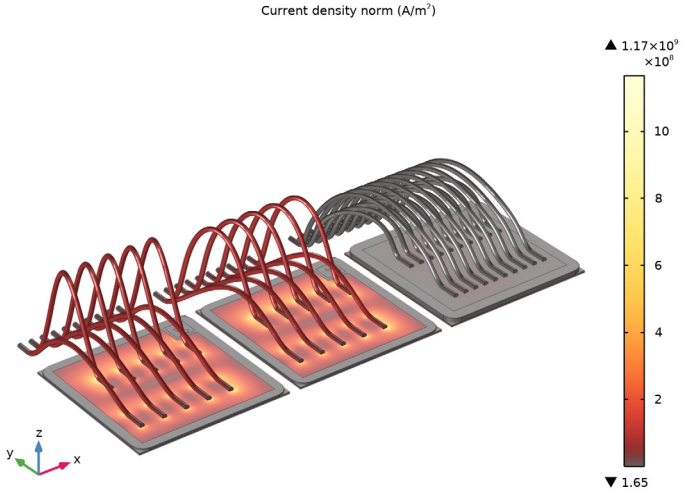
1 In the **Model Builder** window, under **Results** click **Current Density Norm (ec)**.

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

3 From the **View** list, choose **View 3D 2**.

4 In the **Current Density Norm (ec)** toolbar, click **Plot**.

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



Note that only a small portion of the current leaks through the freewheeling diodes.

Temperature (ht)

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

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

3 Locate the **Plot Settings** section. Clear the **Plot dataset edges** checkbox.

4 Locate the **Color Legend** section. Select the **Show maximum and minimum values** checkbox.

5 Locate the **Quality** section. From the **Smoothing** list, choose **Inside geometry domains**.

Surface 1

1 Right-click **Temperature (ht)** and choose **Surface**.

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

3 In the **Expression** text field, type `T`.

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

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

6 From the **Color table transformation** list, choose **Nonlinear**.

7 In the **Color calibration parameter** text field, type `-0.8`.

8 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

Line 1

1 In the **Model Builder** window, right-click **Temperature (ht)** and choose **Line**.

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

3 In the **Expression** text field, type `0`.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

5 Locate the **Coloring and Style** section. From the **Coloring** list, choose **Uniform**.

6 From the **Color** list, choose **Black**.

7 Locate the **Quality** section. From the **Evaluation settings** list, choose **Manual**.

Transparency 1

1 Right-click **Line 1** and choose **Transparency**.

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

3 Find the **Transparency** subsection. In the **Transparency** text field, type `0.75`.

Temperature (ht)

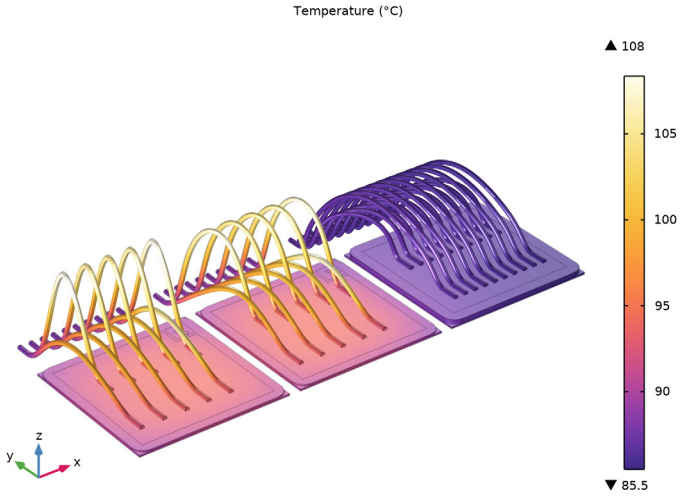
1 In the **Model Builder** window, under **Results** click **Temperature (ht)**.

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

3 From the **View** list, choose **View 3D 2**.

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

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



The plot shows that the highest temperatures are reached in the bond wires.

Current Density Norm (ec), Electric Potential (ec), Temperature (ht)

Now, group the plots for better readability.

1 In the **Model Builder** window, under **Results**, Ctrl-click to select **Electric Potential (ec)**, **Current Density Norm (ec)**, and **Temperature (ht)**.

2 Right-click and choose **Group**.

Set of Dies

In the **Settings** window for **Group**, type `Set of Dies` in the **Label** text field.

The following steps will help you to set up a new set of plots, similar to the previous ones, but showing larger parts of the module.

Module Section

1 Right-click **Set of Dies** and choose **Duplicate**.

2 In the **Model Builder** window, click **Set of Dies 1**.

3 In the **Settings** window for **Group**, type `Module Section` in the **Label** text field.

Electric Potential (ec) 1

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

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 In the **Electric Potential (ec) 1** toolbar, click **Plot**.

View 3D 3

In the **Model Builder** window, under **Results** right-click **Views** and choose **View 3D**.

Camera

1 Click the **Show Grid** button in the **Graphics** toolbar.

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

3 In the **Model Builder** window, expand the **View 3D 3** node, then click **Camera**.

4 In the **Settings** window for **Camera**, click **Update**.

Electric Potential (ec) 1

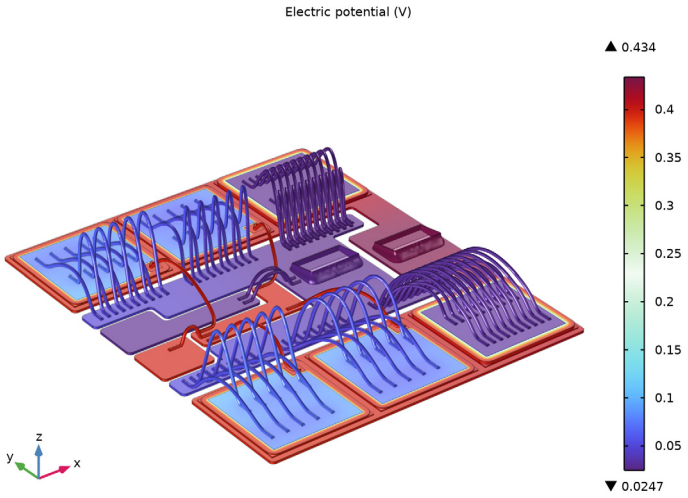
1 In the **Model Builder** window, under **Results** > **Module Section** click **Electric Potential (ec) 1**.

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

3 From the **View** list, choose **View 3D 3**.

4 In the **Electric Potential (ec) 1** toolbar, click **Plot**.

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



Current Density Norm (ec) 1

1 In the **Model Builder** window, click **Current Density Norm (ec) 1**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 In the **Model Builder** window, expand the **Current Density Norm (ec) 1** node.

Selection 1

1 In the **Model Builder** window, expand the **Results** > **Module Section** > **Current Density Norm (ec) 1** > **Surface 1** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Module Section (Not Metalization)**.

Selection 1

1 In the **Model Builder** window, expand the **Results** > **Module Section** > **Current Density Norm (ec) 1** > **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Module Section (Metalization)**.

Current Density Norm (ec) 1

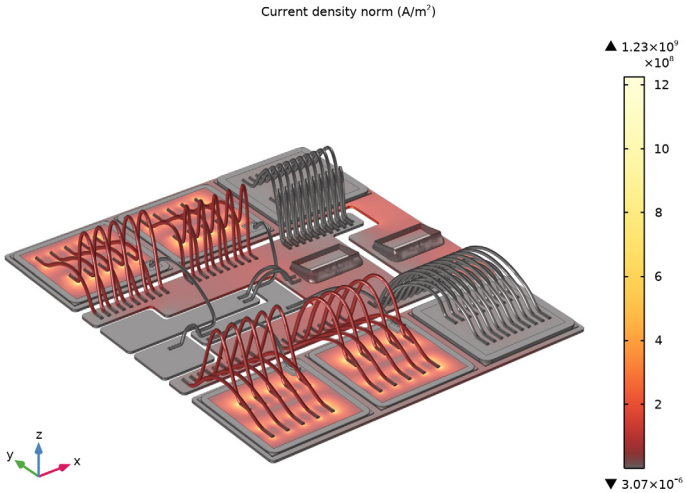
1 In the **Model Builder** window, under **Results** > **Module Section** click **Current Density Norm (ec) 1**.

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

3 From the **View** list, choose **View 3D 3**.

4 In the **Current Density Norm (ec) 1** toolbar, click **Plot**.

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



The figure suggests that the highest current densities occur in proximity to the bonding wires' attachment points. This explains why these areas are of interest when it comes to analyzing the long term effects of heat generation and temperature in an IGBT module.

Temperature (ht) 1

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

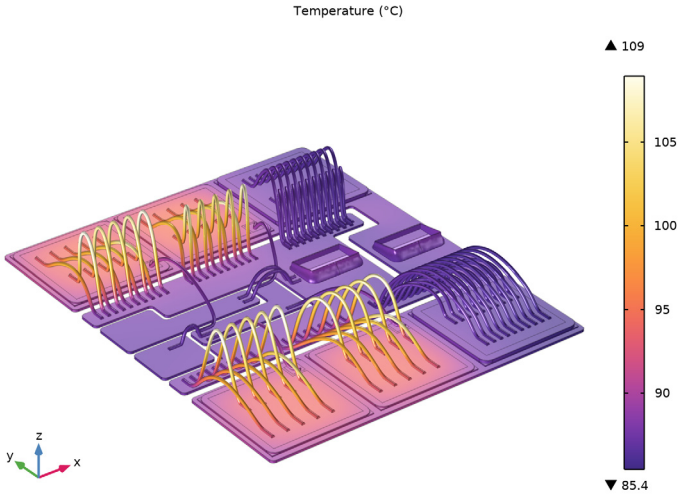
2 In the **Settings** window for **3D Plot Group**, locate the **Data** section.

3 From the **Dataset** list, choose **Study 1/Solution 1 (2) (sol1)**.

4 Locate the **Plot Settings** section. From the **View** list, choose **View 3D 3**.

5 In the **Temperature (ht) 1** toolbar, click **Plot**.

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



All Domains

1 In the **Model Builder** window, right-click **Set of Dies** and choose **Duplicate**.

2 In the **Settings** window for **Group**, type `All Domains` in the **Label** text field.

Electric Potential (ec) 2

1 In the **Model Builder** window, expand the **All Domains** node, then click **Electric Potential (ec) 2**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

Selection 1

1 In the **Model Builder** window, expand the **Electric Potential (ec) 2** node.

2 Right-click **Surface 1** and choose **Selection**.

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

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

5 From the **Selection** list, choose **Electric Current Domains**.

Selection 1

1 In the **Model Builder** window, expand the **Results** > **All Domains** > **Electric Potential (ec) 2** > **Line 1** node.

2 Right-click **Line 1** and choose **Selection**.

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

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

5 From the **Selection** list, choose **Heat Transfer Domains**.

Electric Potential (ec) 2

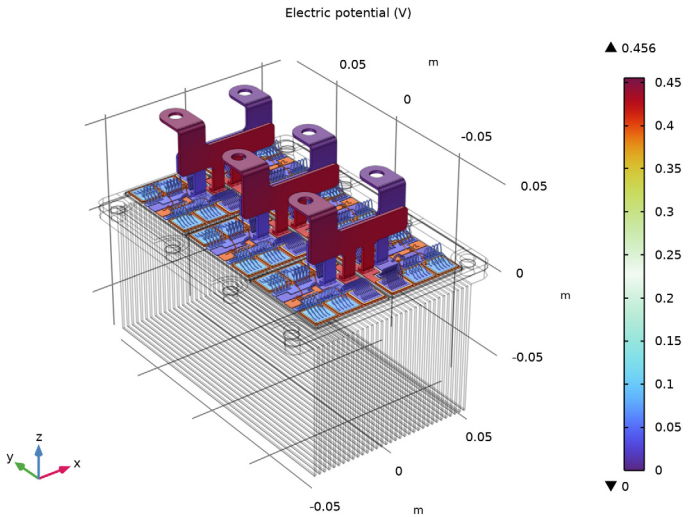
1 In the **Model Builder** window, under **Results** > **All Domains** click **Electric Potential (ec) 2**.

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

3 From the **View** list, choose **Automatic**.

4 In the **Electric Potential (ec) 2** toolbar, click **Plot**.

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



Current Density Norm (ec) 2

1 In the **Model Builder** window, click **Current Density Norm (ec) 2**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

4 In the **Model Builder** window, expand the **Current Density Norm (ec) 2** node.

Selection 1

1 In the **Model Builder** window, expand the **Results** > **All Domains** > **Current Density Norm (ec) 2** > **Surface 1** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **All Domains (EC, Not Metalization)**.

Selection 1

1 In the **Model Builder** window, expand the **Results** > **All Domains** > **Current Density Norm (ec) 2** > **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **All Domains (EC, Metalization)**.

Selection 1

1 In the **Model Builder** window, expand the **Results** > **All Domains** > **Current Density Norm (ec) 2** > **Line 1** node.

2 Right-click **Line 1** and choose **Selection**.

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

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

5 From the **Selection** list, choose **Heat Transfer Domains**.

Current Density Norm (ec) 2

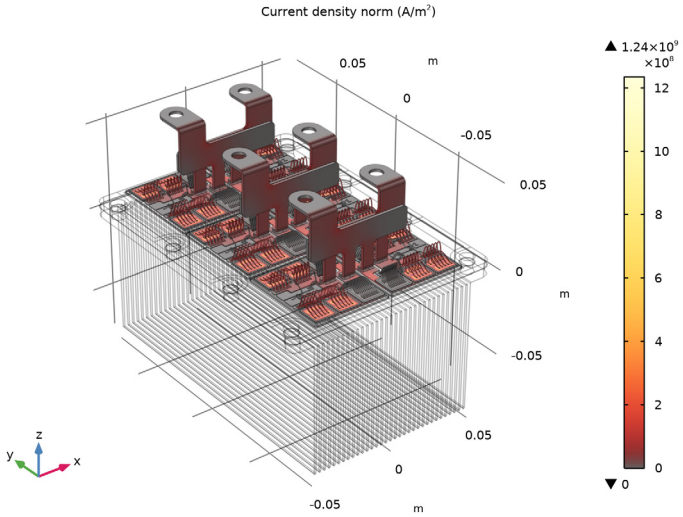
1 In the **Model Builder** window, under **Results** > **All Domains** click **Current Density Norm (ec) 2**.

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

3 From the **View** list, choose **Automatic**.

4 In the **Current Density Norm (ec) 2** toolbar, click **Plot**.

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



Temperature (ht) 2

1 In the **Model Builder** window, click **Temperature (ht) 2**.

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

3 From the **Dataset** list, choose **Study 1/Solution 1 (3) (sol1)**.

Selection 1

1 In the **Model Builder** window, expand the **Temperature (ht) 2** node.

2 Right-click **Surface 1** and choose **Selection**.

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

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

5 From the **Selection** list, choose **Heat Transfer Domains**.

Surface 2

1 Right-click **Surface 1** and choose **Duplicate**.

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

3 In the **Expression** text field, type 0.

4 Locate the **Title** section. From the **Title type** list, choose **None**.

Selection 1

1 In the **Model Builder** window, expand the **Surface 2** node, then click **Selection 1**.

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

3 From the **Selection** list, choose **Plastic Enclosure (Import 1)**.

Filter 1

1 In the **Model Builder** window, right-click **Surface 2** and choose **Filter**.

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

3 In the **Logical expression for inclusion** text field, type `y>0.045`.

Material Appearance 1

1 Right-click **Surface 2** and choose **Material Appearance**.

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

3 From the **Appearance** list, choose **Custom**.

4 From the **Color** list, choose **Black**.

Surface 3

Right-click **Surface 2** and choose **Duplicate**.

Filter 1

1 In the **Model Builder** window, expand the **Surface 3** node, then click **Filter 1**.

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

3 In the **Logical expression for inclusion** text field, type `x>0.022`.

Selection 1

1 In the **Model Builder** window, expand the **Results** > **All Domains** > **Temperature (ht) 2** > **Line 1** node.

2 Right-click **Line 1** and choose **Selection**.

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

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

5 From the **Selection** list, choose **Heat Transfer Domains**.

Temperature (ht) 2

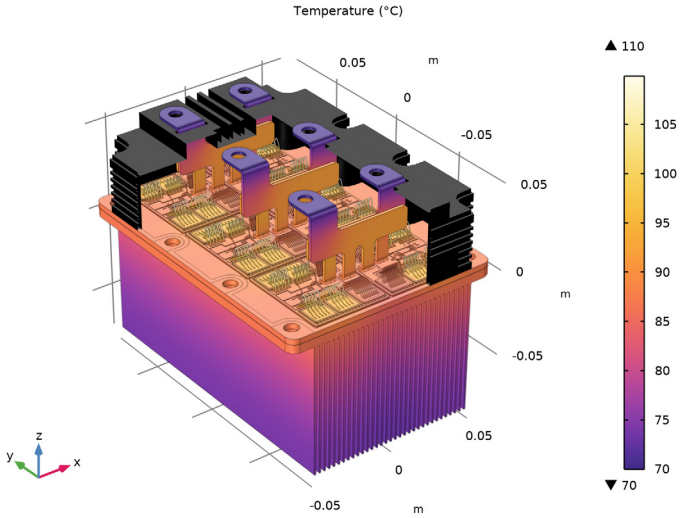
1 In the **Model Builder** window, under **Results** > **All Domains** click **Temperature (ht) 2**.

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

3 From the **View** list, choose **Automatic**.

4 In the **Temperature (ht) 2** toolbar, click **Plot**.

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



Evaluation Group 1

In the **Results** toolbar, click **Evaluation Group**.

Global Evaluation 1

1 Right-click **Evaluation Group 1** and choose **Global Evaluation**.

2 In the **Settings** window for **Global Evaluation**, locate the **Expressions** section.

3 In the table, enter the following settings:

Expression	Unit	Description
J_igbt	kA/m^2	Current density across junction (average)
T_igbt	degC	Junction temperature (average)
V_igbt	mV	Collector-emitter voltage IGBT (open, approximate)
s_igbt	S/m	Effective conductivity IGBT (open, approximate)

Evaluation Group 1

1 In the **Model Builder** window, click **Evaluation Group 1**.

2 In the **Settings** window for **Evaluation Group**, locate the **Transformation** section.

3 Select the **Transpose** checkbox.

4 In the **Evaluation Group 1** toolbar, click = **Evaluate**.

This should result in a current density of 526 kA/m^2, a junction temperature of 93.5°C, a voltage drop of 324 mV, and a conductivity of 227 S/m. Your values may vary slightly. The derived conductivity for the semiconductor explains why a majority of the current passes through the IGBT chips instead of the FWDs which, for reference, had a conductivity defined at 10 S/m. Note also that the collector–emitter voltage makes up the majority of the total voltage drop over the current path.

Appendix

The following steps show how to create the geometry, the selections, and the mesh. The geometry is rather involved and is not the main focus of this tutorial. Thus, it is imported from a CAD file. Geometry selections are made based on the output objects of the file.

From the **File** menu, choose **New**.

NEW

In the **New** window, click **Model Wizard**.

MODEL WIZARD

1 In the **Model Wizard** window, click **3D**.

For this introductory model, we will not select any physics.

2 Click **Done**.

GLOBAL DEFINITIONS

Start by adding global parameters. These are used later for setting up and generalizing the physics.

Parameters 1

1 In the **Model Builder** window, under **Global Definitions** click **Parameters 1**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
I_col	1.8[kA]	1800 A	Rated collector current (entire module)
th_met	4[um]	4E-6 m	Metalization layer thickness (diode and IGBT die)
th_igbt	140[um]	1.4E-4 m	IGBT die thickness
h_air	50[W/(m^2*K)]	50 W/(m²·K)	Heat transfer coefficient, air (forced convection)
T_air	60[degC]	333.15 K	Ambient temperature, air (forced convection)
T_trm	70[degC]	343.15 K	Ambient temperature, terminals (conduction)

The metalization thickness is used in boundary layer features and the IGBT die thickness is used to calculate the effective conductivity.

GEOMETRY 1

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

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

3 Clear the **Automatic detection of small details** checkbox.

4 Locate the **Advanced** section. From the **Geometry representation** list, choose **CAD kernel**.

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

Import 1 (imp1)

Import the geometry from `igbt_joule_heating_geom.step`.

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

2 In the **Settings** window for **Import**, locate the **Source** section.

3 From the **Source** list, choose **3D CAD file**.

4 Click **Browse**.

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

6 Click **Import**.

The imported CAD file already contains many selections for this model. These can be found in the **Import** settings window. Inspect these selections and provide labels for them.

7 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

8 Click to expand the **Object Selections** section. Select the **Show names from file** checkbox.

9 In the table, enter the following settings:

Name	Name in file	Keep	Physics	Contribute to
IGBT Die	Color 1	√	√	None
Diode Die	Color 2	√	√	None
Plastic Enclosure	Color 3	√	√	None
Bond Wires	Color 4	√	√	None
Heat Sink	Color 5	√	√	None
DCB Bottom Cu Layer	Color 6	√	√	None
DCB Upper Cu Layer	Color 7	√	√	None
Collector Busbar	Color 8	√	√	None
Emitter Busbar	Color 9	√	√	None
Backing Plate	Color 10	√	√	None
DCB Aluminum Oxide Layer	Color 11	√	√	None
DCB Solder	Color 12	√	√	None
Diode Solder	Color 13	√	√	None
IGBT Solder	Color 14	√	√	None
Busbar Solder	Color 15	√	√	None
Thermal Paste	Color 16	√	√	None

Next, do the same for the five boundary selections.

10 Click to expand the **Boundary Selections** section. Select the **Show names from file** checkbox.

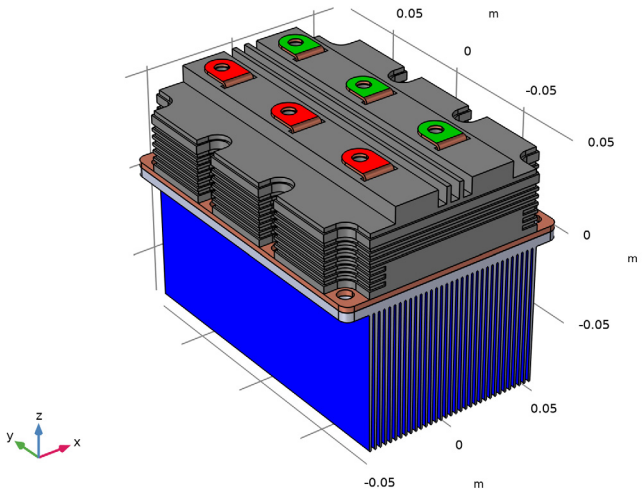
11 In the table, enter the following settings:

Name	Name in file	Keep	Physics	Contribute to
`Forced Convection`	Boundary color 1	√	√	None
`Ground`	Boundary color 2	√	√	None
`IGBT Upper Electrode`	Boundary color 3	√	√	None
`Remaining Electrodes`	Boundary color 4	√	√	None
`Terminal`	Boundary color 5	√	√	None

Disable the wireframe to make the assigned colors visible.

12 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

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



IGBT Integration Domain

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

2 In the **Settings** window for **Extrude**, type `IGBT Integration Domain` in the **Label** text field.

3 Locate the **General** section. From the **Input faces** list, choose **IGBT Upper Electrode (Import 1)**.

4 From the **Input object handling** list, choose **Keep**.

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

Distances (m)
th_igbt

6 Select the **Reverse direction** checkbox.

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

8 Locate the **Selections on Input Objects** section. Clear the **Propagate selections to resulting objects** checkbox.

Form Union (fin)

In the **Geometry** toolbar, click **Build All**, this might take a few minutes.

The following instructions show how to create geometry selections for the model. This will simplify the workflow considerably.

Aluminum (Metalization)

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

2 In the **Settings** window for **Union Selection**, type `Aluminum (Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog, in the **Selections to add** list, choose **IGBT Upper Electrode (Import 1)** and **Remaining Electrodes (Import 1)**.

6 Click **OK**.

Copper

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

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

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

4 In the **Add** dialog, in the **Selections to add** list, choose **DCB Bottom Cu Layer (Import 1)**, **DCB Upper Cu Layer (Import 1)**, **Collector Busbar (Import 1)**, **Emitter Busbar (Import 1)**, and **Backing Plate (Import 1)**.

5 Click **OK**.

Solder (SnAgCu305)

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

2 In the **Settings** window for **Union Selection**, type `Solder (SnAgCu305)` in the **Label** text field.

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

4 In the **Add** dialog, in the **Selections to add** list, choose **DCB Solder (Import 1)**, **Diode Solder (Import 1)**, **IGBT Solder (Import 1)**, and **Busbar Solder (Import 1)**.

5 Click **OK**.

Aluminum

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

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

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

4 In the **Add** dialog, in the **Selections to add** list, choose **Bond Wires (Import 1)** and **Heat Sink (Import 1)**.

5 Click **OK**.

Aluminum (Metalization) (unisel1) — Aluminum (unisel4)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, ctrl-click to select all nodes from **Aluminum (Metalization) (unisel1)** to **Aluminum (unisel4)**.

2 Right-click and choose **Group**.

Materials Selections

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

2 In the **Settings** window for **Group**, type `Materials Selections` in the **Label** text field.

3 In the **Model Builder** window, collapse the **Materials Selections** node.

Electric Current Domains

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

2 In the **Settings** window for **Union Selection**, type `Electric Current Domains` in the **Label** text field.

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

4 In the **Add** dialog, in the **Selections to add** list, choose **IGBT Die (Import 1)**, **Diode Die (Import 1)**, **Bond Wires (Import 1)**, **DCB Upper Cu Layer (Import 1)**, **Collector Busbar (Import 1)**, **Emitter Busbar (Import 1)**, **Diode Solder (Import 1)**, **IGBT Solder (Import 1)**, and **Busbar Solder (Import 1)**.

5 Click **OK**.

Heat Transfer Domains

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

2 In the **Settings** window for **Complement Selection**, type `Heat Transfer Domains` in the **Label** text field.

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

4 In the **Add** dialog, select **Plastic Enclosure (Import 1)** in the **Selections to invert** list.

5 Click **OK**.

Fixed Temperature

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

2 In the **Settings** window for **Union Selection**, type `Fixed Temperature` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog, in the **Selections to add** list, choose **Ground (Import 1)** and **Terminal (Import 1)**.

6 Click **OK**.

Electric Current Domains (unisel5) — Fixed Temperature (unisel6)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, ctrl-click to select all nodes from **Electric Current Domains (unisel5)** to **Fixed Temperature (unisel6)**.

2 Right-click and choose **Group**.

Physics Selections

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

2 In the **Settings** window for **Group**, type `Physics Selections` in the **Label** text field.

3 In the **Model Builder** window, collapse the **Physics Selections** node.

DCB Base Layers

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

2 In the **Settings** window for **Union Selection**, type `DCB Base Layers` in the **Label** text field.

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

4 In the **Add** dialog, in the **Selections to add** list, choose **DCB Bottom Cu Layer (Import 1)**, **DCB Aluminum Oxide Layer (Import 1)**, and **DCB Solder (Import 1)**.

5 Click **OK**.

Busbars

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

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

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

4 In the **Add** dialog, in the **Selections to add** list, choose **Collector Busbar (Import 1)** and **Emitter Busbar (Import 1)**.

5 Click **OK**.

IGBT and Diode Solder

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

2 In the **Settings** window for **Union Selection**, type `IGBT and Diode Solder` in the **Label** text field.

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

4 In the **Add** dialog, in the **Selections to add** list, choose **Diode Solder (Import 1)** and **IGBT Solder (Import 1)**.

5 Click **OK**.

IGBT and Diode Die

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

2 In the **Settings** window for **Union Selection**, type `IGBT and Diode Die` in the **Label** text field.

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

4 In the **Add** dialog, in the **Selections to add** list, choose **IGBT Die (Import 1)** and **Diode Die (Import 1)**.

5 Click **OK**.

IGBT and Diode Die and Solder

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

2 In the **Settings** window for **Union Selection**, type `IGBT and Diode Die and Solder` in the **Label** text field.

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

4 In the **Add** dialog, in the **Selections to add** list, choose **IGBT and Diode Solder** and **IGBT and Diode Die**.

5 Click **OK**.

Busbars and Solder

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

2 In the **Settings** window for **Union Selection**, type `Busbars and Solder` in the **Label** text field.

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

4 In the **Add** dialog, in the **Selections to add** list, choose **DCB Upper Cu Layer (Import 1)**, **Busbar Solder (Import 1)**, and **Busbars**.

5 Click **OK**.

Swept 2

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

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

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

4 In the **Add** dialog, in the **Selections to add** list, choose **Thermal Paste (Import 1)** and **DCB Base Layers**.

5 Click **OK**.

Adjacent Selection 1 (adjsel1)

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

2 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.

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

4 Click **Add**.

5 In the **Add** dialog, select **Bond Wires (Import 1)** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent Selection**, locate the **Resulting Selection** section.

8 From the **Show in physics** list, choose **Off**.

Adjacent Selection 2 (adjsel2)

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

2 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.

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

4 Click **Add**.

5 In the **Add** dialog, select **Adjacent Selection 1** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent Selection**, locate the **Resulting Selection** section.

8 From the **Show in physics** list, choose **Off**.

Adjacent Selection 3 (adjsel3)

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

2 In the **Settings** window for **Adjacent Selection**, locate the **Input Entities** section.

3 Click **Add**.

4 In the **Add** dialog, select **IGBT and Diode Die** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Resulting Selection** section.

7 From the **Show in physics** list, choose **Off**.

Free Triangular 1

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

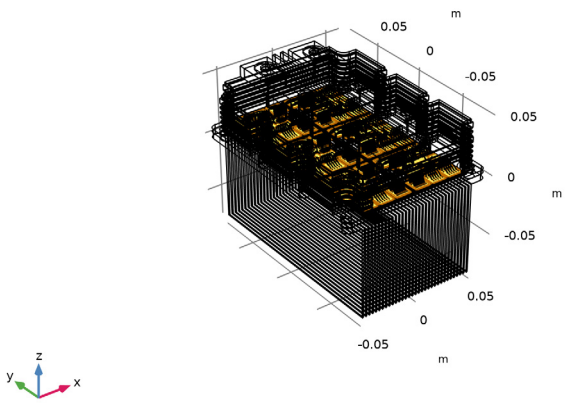
2 In the **Settings** window for **Logical Expression Selection**, type `Free Triangular 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Expression** section. In the **Logical expression** text area, type `adjsel1 || (adjsel2 && adjsel3 && !imp1_Color_4)`.

5 Click the **Wireframe Rendering** button in the **Graphics** toolbar.

Compare the selection with the figure below.



Boundary Layers 1

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

2 In the **Settings** window for **Difference Selection**, type `Boundary Layers 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Free Triangular 1** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Adjacent Selection 1** in the **Selections to subtract** list.

10 Click **OK**.

Boundary Layer Properties

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

2 In the **Settings** window for **Adjacent Selection**, type `Boundary Layer Properties` in the **Label** text field.

3 Locate the **Input Entities** section. From the **Geometric entity level** list, choose **Boundary**.

4 Click **Add**.

5 In the **Add** dialog, select **Free Triangular 1** in the **Input selections** list.

6 Click **OK**.

7 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

8 From the **Geometric entity level** list, choose **Adjacent edges**.

9 From the **Exterior edges** list, choose **None**.

10 Select the **Interior edges** checkbox.

Free Tetrahedral 3: Size 1

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

2 In the **Settings** window for **Box Selection**, type `Free Tetrahedral 3: Size 1` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Box Limits** section. In the **x minimum** text field, type `-0.05`.

5 In the **x maximum** text field, type `0.05`.

6 In the **y minimum** text field, type `-0.09`.

7 In the **y maximum** text field, type `0.09`.

8 In the **z minimum** text field, type `-0.001`.

9 In the **z maximum** text field, type `0.001`.

10 Locate the **Output Entities** section. From the **Include entity if** list, choose **All vertices inside box**.

DCB Base Layers (unisel7) — Free Tetrahedral 3: Size 1 (boxsel1)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, ctrl-click to select all nodes from **DCB Base Layers (unisel7)** to **Free Tetrahedral 6: Size 2 (boxsel1)**.

2 Right-click and choose **Group**.

Mesh Selections

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Group 3**.

2 In the **Settings** window for **Group**, type `Mesh Selections` in the **Label** text field.

3 In the **Model Builder** window, collapse the **Mesh Selections** node.

Set of Dies (Domain)

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

2 In the **Settings** window for **Box Selection**, type `Set of Dies (Domain)` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `-0.0465`.

4 In the **x maximum** text field, type `0`.

5 In the **y minimum** text field, type `-0.085`.

6 In the **y maximum** text field, type `-0.06`.

7 In the **z minimum** text field, type `0.009`.

8 In the **z maximum** text field, type `0.02`.

9 Locate the **Output Entities** section. From the **Include entity if** list, choose **All vertices inside box**.

Set of Dies (Boundary)

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

2 In the **Settings** window for **Adjacent Selection**, type `Set of Dies (Boundary)` in the **Label** text field.

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

4 In the **Add** dialog, select **Set of Dies (Domain)** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

7 Select the **Interior boundaries** checkbox.

Set of Dies (Not Metalization)

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

2 In the **Settings** window for **Difference Selection**, type `Set of Dies (Not Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Set of Dies (Boundary)** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Aluminum (Metalization)** in the **Selections to subtract** list.

10 Click **OK**.

Set of Dies (Metalization)

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

2 In the **Settings** window for **Intersection Selection**, type `Set of Dies (Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog, in the **Selections to intersect** list, choose **Aluminum (Metalization)** and **Set of Dies (Boundary)**.

6 Click **OK**.

Module Section (Domain)

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

2 In the **Settings** window for **Box Selection**, type `Module Section (Domain)` in the **Label** text field.

3 Locate the **Box Limits** section. In the **x minimum** text field, type `-0.0465`.

4 In the **x maximum** text field, type `0`.

5 In the **y minimum** text field, type `-0.085`.

6 In the **y maximum** text field, type `-0.0275`.

7 In the **z minimum** text field, type `0.009`.

8 In the **z maximum** text field, type `0.02`.

9 Locate the **Output Entities** section. From the **Include entity if** list, choose **All vertices inside box**.

Module Section (Boundary)

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

2 In the **Settings** window for **Adjacent Selection**, type `Module Section (Boundary)` in the **Label** text field.

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

4 In the **Add** dialog, select **Module Section (Domain)** in the **Input selections** list.

5 Click **OK**.

6 In the **Settings** window for **Adjacent Selection**, locate the **Output Entities** section.

7 Select the **Interior boundaries** checkbox.

Module Section (Not Metalization)

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

2 In the **Settings** window for **Difference Selection**, type `Module Section (Not Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **Module Section (Boundary)** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Aluminum (Metalization)** in the **Selections to subtract** list.

10 Click **OK**.

Module Section (Metalization)

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

2 In the **Settings** window for **Intersection Selection**, type `Module Section (Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog, in the **Selections to intersect** list, choose **Aluminum (Metalization)** and **Module Section (Boundary)**.

6 Click **OK**.

All Domains (Not EC)

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

2 In the **Settings** window for **Difference Selection**, type `All Domains (Not EC)` in the **Label** text field.

3 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

4 In the **Add** dialog, select **Heat Transfer Domains** in the **Selections to add** list.

5 Click **OK**.

6 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

7 Click the **Add** button for **Selections to subtract**.

8 In the **Add** dialog, select **Electric Current Domains** in the **Selections to subtract** list.

9 Click **OK**.

All Domains (EC, Boundary)

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

2 In the **Settings** window for **Adjacent Selection**, type `All Domains (EC, Boundary)` in the **Label** text field.

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

4 In the **Add** dialog, select **Electric Current Domains** in the **Input selections** list.

5 Click **OK**.

All Domains (EC, Not Metalization)

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

2 In the **Settings** window for **Difference Selection**, type `All Domains (EC, Not Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

4 Locate the **Input Entities** section. Click the **Add** button for **Selections to add**.

5 In the **Add** dialog, select **All Domains (EC, Boundary)** in the **Selections to add** list.

6 Click **OK**.

7 In the **Settings** window for **Difference Selection**, locate the **Input Entities** section.

8 Click the **Add** button for **Selections to subtract**.

9 In the **Add** dialog, select **Aluminum (Metalization)** in the **Selections to subtract** list.

10 Click **OK**.

All Domains (EC, Metalization)

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

2 In the **Settings** window for **Intersection Selection**, type `All Domains (EC, Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Level** section. From the **Level** list, choose **Boundary**.

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

5 In the **Add** dialog, in the **Selections to intersect** list, choose **Aluminum (Metalization)** and **All Domains (EC, Boundary)**.

6 Click **OK**.

Set of Dies (Domain) (boxsel2) — All Domains (EC, Metalization) (intsel3)

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1**, ctrl-click to select all nodes from **Set of Dies (Domain) (boxsel2)** to **All Domains (EC, Metalization) (intsel3)**.

2 Right-click and choose **Group**.

Results Selections

1 In the **Model Builder** window, under **Component 1 (comp1)** > **Geometry 1** click **Group 4**.

2 In the **Settings** window for **Group**, type `Results Selections` in the **Label** text field.

GEOMETRY 1

1 In the **Model Builder** window, collapse the **Component 1 (comp1)** > **Geometry 1** > **Results Selections** node.

With this step, all selections are now in order. Continue by removing unnecessary details and hiding the plastic container to get a better view of the geometry.

Remove Details 1 (rmd1)

1 In the **Geometry** toolbar, click **Virtual Operations** and choose **Remove Details**.

2 In the **Settings** window for **Remove Details**, locate the **Entities to Process** section.

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

4 From the **Selection** list, choose **Bond Wires (Import 1)**.

5 Locate the **Details to Remove** section. Clear the **Vertices with continuous tangent** checkbox.

6 Clear the **Short edges** checkbox.

7 Clear the **Small faces** checkbox.

8 Clear the **Sliver faces** checkbox.

9 Clear the **Thin domains** checkbox.

10 Locate the **Parameters** section. From the **Detail size** list, choose **Absolute**.

11 In the **Maximum absolute size** text field, type `1.70005E-5`.

12 Click **Build Selected**.

DEFINITIONS

View 1

In the **Model Builder** window, expand the **Component 1 (comp1)** > **Definitions** node, then click **View 1**.

Hide for Geometry 1

1 In the **View 1** toolbar, click **Hide**.

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

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

4 From the **Selection** list, choose **Plastic Enclosure (Import 1)**.

MATERIALS

Next, assign materials to the different selections. The material properties will be defined later after having set up the physics.

Solder (SnAgCu305)

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

2 In the **Settings** window for **Material**, type `Solder (SnAgCu305)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Solder (SnAgCu305)**.

Copper (Internal)

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

2 In the **Settings** window for **Material**, type `Copper (Internal)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Copper**.

Aluminium (Metalization)

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

2 In the **Settings** window for **Material**, type `Aluminium (Metalization)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.

4 From the **Selection** list, choose **Aluminum (Metalization)**.

Diode (Closed)

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

2 In the **Settings** window for **Material**, type `Diode (Closed)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Diode Die (Import 1)**.

IGBT (Open)

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

2 In the **Settings** window for **Material**, type `IGBT (Open)` in the **Label** text field.

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **IGBT Die (Import 1)**.

Aluminum

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Aluminum**.

Aluminum Oxide

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **DCB Aluminum Oxide Layer (Import 1)**.

Thermal Paste

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

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

3 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Thermal Paste (Import 1)**.

MESH 1

Next, create the mesh. It needs to take into consideration the large variation between different scales of the model and what domains are of interest.

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

2 In the **Settings** window for **Mesh**, locate the **Sequence Type** section.

3 From the list, choose **User-controlled mesh**.

Size

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

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

3 In the **Minimum element size** text field, type `0.1288[mm]`.

Size 1

In the **Model Builder** window, right-click **Size 1** and choose **Delete**.

Free Tetrahedral 1

1 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

2 From the **Geometric entity level** list, choose **Domain**.

3 From the **Selection** list, choose **Bond Wires (Import 1)**.

Size 1

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

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

3 Click the **Custom** button.

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

5 Select the **Minimum element size** checkbox. In the associated text field, type `0.009[mm]`.

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 **Free Triangular 1**.

Size 1

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

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** checkbox. In the associated text field, type `1[mm]`.

6 Select the **Maximum element growth rate** checkbox. In the associated text field, type `1.15`.

Boundary Layers 1

1 In the **Mesh** toolbar, click  **Boundary Layers**.

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

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

4 From the **Selection** list, choose **Boundary Layers 1**.

Boundary Layer Properties

1 In the **Model Builder** window, click **Boundary Layer Properties**.

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

3 From the **Selection** list, choose **Boundary Layer Properties**.

4 Locate the **Layers** section. In the **Number of layers** text field, type `3`.

5 From the **Thickness specification** list, choose **First layer**.

6 In the **Thickness** text field, type `0.05[mm]`.

Swept 1

1 In the **Mesh** toolbar, click  **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

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

4 From the **Selection** list, choose **IGBT and Diode Die and Solder**.

5 Locate the **Mesh Generation** section. From the **Elements** list, choose **Prisms**.

Distribution 1

1 In the **Mesh** toolbar, click  **Distribution**.

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

3 From the **Distribution type** list, choose **Predefined**.

4 In the **Number of elements** text field, type `3`.

5 In the **Element ratio** text field, type `3`.

Distribution 2

1 In the **Mesh** toolbar, click  **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **IGBT and Diode Solder**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `2`.

Free Tetrahedral 2

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

2 In the **Settings** window for **Free Tetrahedral**, locate the **Domain Selection** section.

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

4 From the **Selection** list, choose **Busbars and Solder**.

Size 1

1 Right-click **Free Tetrahedral 2** and choose **Size**.

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

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

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

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

6 Select the **Maximum element size** checkbox. In the associated text field, type `1[mm]`.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 2** and choose **Size**.

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

3 From the **Selection** list, choose **Busbar Solder (Import 1)**.

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

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

6 Select the **Maximum element growth rate** checkbox. In the associated text field, type `2`.

7 Select the **Curvature factor** checkbox. In the associated text field, type `0.8`.

Swept 2

1 In the **Mesh** toolbar, click **Swept**.

2 In the **Settings** window for **Swept**, locate the **Domain Selection** section.

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

4 From the **Selection** list, choose **Swept 2**.

5 Locate the **Mesh Generation** section. From the **Elements** list, choose **Prisms**.

Size 1

1 Right-click **Swept 2** and choose **Size**.

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

3 From the **Selection** list, choose **DCB Base Layers**.

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

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

6 Select the **Maximum element size** checkbox. In the associated text field, type `1[mm]`.

7 Select the **Minimum element size** checkbox. In the associated text field, type `0.4[mm]`.

Size 2

1 In the **Model Builder** window, right-click **Swept 2** and choose **Size**.

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

3 Click the **Custom** button.

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

5 Select the **Maximum element size** checkbox. In the associated text field, type `3[mm]`.

6 Locate the **Geometric Entity Selection** section. From the **Selection** list, choose **Thermal Paste (Import 1)**.

Distribution 1

1 In the **Mesh** toolbar, click  **Distribution**.

2 In the **Settings** window for **Distribution**, locate the **Domain Selection** section.

3 From the **Selection** list, choose **DCB Base Layers**.

4 Locate the **Distribution** section. In the **Number of elements** text field, type `3`.

Free Tetrahedral 3

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

Size 1

1 Right-click **Free Tetrahedral 3** 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 From the **Selection** list, choose **Free Tetrahedral 3: Size 1**.

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

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

7 Select the **Minimum element size** checkbox. In the associated text field, type `0.5[mm]`.

8 Select the **Curvature factor** checkbox. In the associated text field, type `0.1`.

Size 2

1 In the **Model Builder** window, right-click **Free Tetrahedral 3** and choose **Size**.

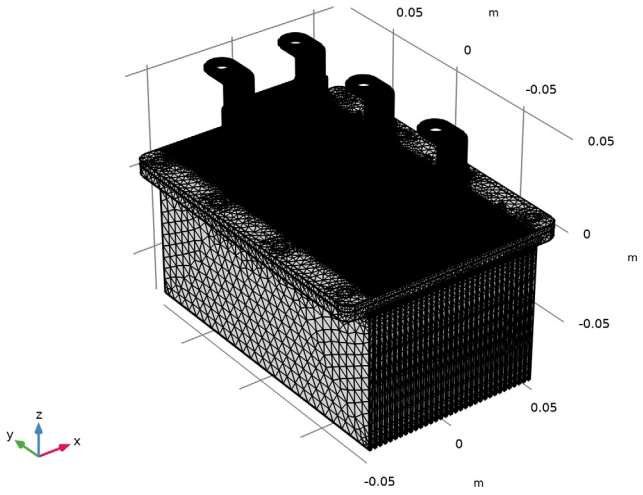
2 In the **Settings** window for **Size**, locate the **Geometric Entity Selection** section.

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

4 From the **Selection** list, choose **Heat Sink (Import 1)**.

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

6 In the **Model Builder** window, right-click **Mesh 1** and choose **Build All**, this might take a few minutes.



You have now finished building the geometry and generating the mesh. Continue to Modeling Instructions to set up the physics and run the study.