Heat Sink

Introduction

This model is intended as a first introduction to simulations of fluid flow and conjugate heat transfer. It shows the following important points:

- How to draw an air box around a device in order to model convective cooling in this box.
- How to set a total heat flux on a boundary using automatic area computation.
- How to display results in an efficient way using selections in data sets.

The model is also described in detail in the book *Introduction to the Heat Transfer Module*. An extension of the model that takes surface-to-surface radiation into account is also available; see Heat Sink with Surface-to-Surface Radiation.

Model Definition

The modeled system consists of an aluminum heat sink for cooling of components in electronic circuits mounted inside a channel of rectangular cross section (see Figure 1). Such a set-up is used to measure the cooling capacity of heat sinks. Air enters the channel at the inlet and exits the channel at the outlet. The base surface of the heat sink receives a 1 W heat flux from an external heat source. All other external faces are thermally insulated.

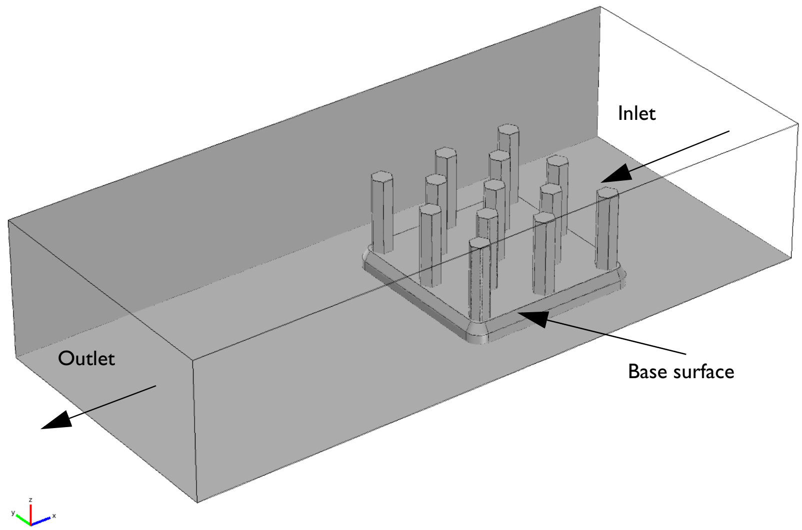


Figure 1: The model set-up including channel and heat sink.

The cooling capacity of the heat sink can be determined by monitoring the temperature of the base surface of the heat sink.

The model solves a thermal balance for the heat sink and the air flowing in the rectangular channel. Thermal energy is transported through conduction in the aluminum heat sink and through conduction and convection in the cooling air. The temperature field is continuous across the internal surfaces between the heat sink and the air in the channel. The temperature is set at the inlet of the channel. The base of the heat sink receives a 1 W heat flux. The transport of thermal energy at the outlet is dominated by convection.

The flow field is obtained by solving one momentum balance for each space coordinate (x, y, and z) and a mass balance. The inlet velocity is defined by a parabolic velocity profile for fully developed laminar flow. At the outlet, the normal stress is equal the outlet pressure and the tangential stress is canceled. At all solid surfaces, the velocity is set to zero in all three spatial directions.

The thermal conductivity of air, the heat capacity of air, and the air density are all temperature-dependent material properties.

You can find all of the settings mentioned above in the physics interface for Conjugate Heat Transfer in COMSOL Multiphysics. You also find the material properties, including their temperature dependence, in the Material Browser.

Results

In Figure 2, the hot wake behind the heat sink visible in the plot is a sign of the convective cooling effects. The maximum temperature, reached at the heat sink base, is slightly less than 379 K.

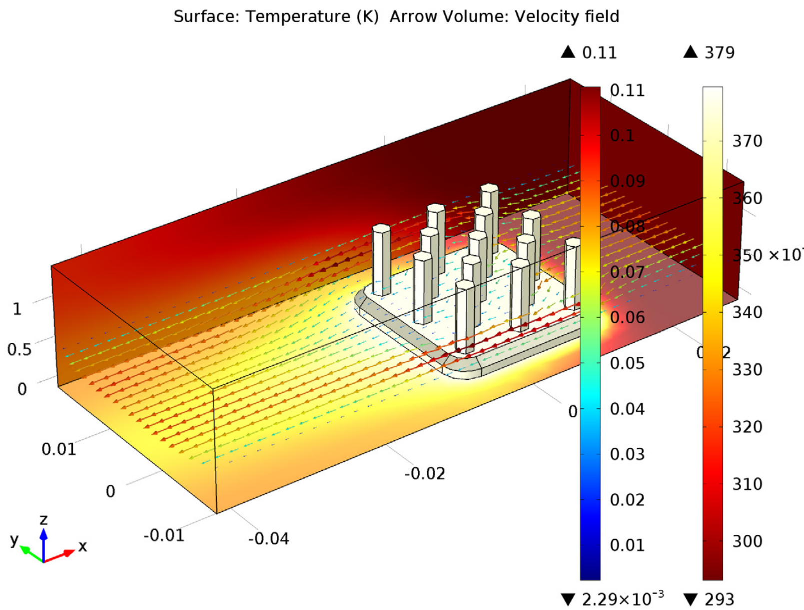


Figure 2: The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.

Model Library path: `Heat_Transfer_Module/`
`Tutorial_Models,_Forced_and_Natural_Convection/heat_sink`

Modeling Instructions

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

NEW

1 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>Conjugate Heat Transfer>Laminar Flow**.
3 Click **Add**.
4 Click **Study**.
5 In the **Select study** tree, select **Preset Studies for Selected Physics Interfaces>Stationary**.
6 Click **Done**.

GEOMETRY 1

The model geometry is available as a parameterized geometry sequence in a separate MPH-file. If you want to build it from scratch, follow the instructions in the section Appendix: Geometry Modeling Instructions. Otherwise load it from file with the following steps.

1 On the **Geometry** toolbar, click **Insert Sequence**.
2 Browse to the model's Model Library folder and double-click the file `heat_sink_geom_sequence.mph`.

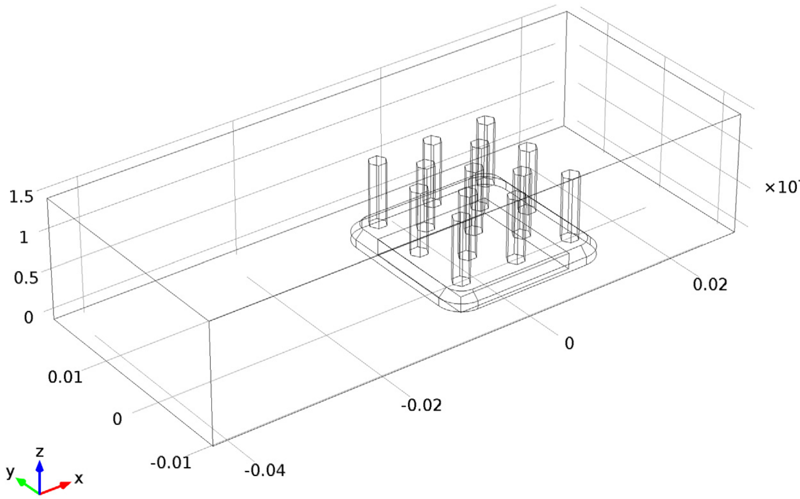
   The model's Model Library folder is shown in the **Model Library path** section immediately before the current section. Note that the path given there is relative to the COMSOL Model Library root, which for a standard installation on Windows is `C:\Program Files\COMSOL\COMSOL50\models`.

Import 1 (imp1)

1 Click the **Go to Default 3D View** button on the **Graphics** toolbar.
2 On the **Geometry** toolbar, click **Build All**.

   To facilitate face selection in the next steps, use the **Wireframe rendering** option (skip this step if you followed the instructions in the appendix):

3 Click the **Wireframe Rendering** button on the **Graphics** toolbar.



LAMINAR FLOW (SPF)

Create a selection for the air domain used the physics interfaces to define the fluid.

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Laminar Flow (spf)**.
2. In the **Settings** window for Laminar Flow, locate the **Domain Selection** section.
3. In the **Selection** list, choose **2** and **3**.
4. Click **Remove from Selection**.
5. Click **Create Selection**.
6. In the **Create Selection** dialog box, type `Air` in the **Selection name** text field.
7. Click **OK**.

HEAT TRANSFER (HT)

On the **Physics** toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Heat Transfer in Fluids 1

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

2 In the **Settings** window for Heat Transfer in Fluids, locate the **Domain Selection** section.

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

MATERIALS

Next, add materials.

ADD MATERIAL

1 On the **Model** toolbar, click **Add Material** to open the **Add Material** window.

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

3 In the tree, select **Built-In>Air**.

4 Click **Add to Component** in the window toolbar.

MATERIALS

Air (mat1)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Air (mat1)**.

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

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

ADD MATERIAL

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

2 In the tree, select **Built-In>Aluminum 3003-H18**.

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

MATERIALS

Aluminum 3003-H18 (mat2)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum 3003-H18 (mat2)**.

2 Select Domain 2 only.

ADD MATERIAL

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

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

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

MATERIALS

Silica glass (mat3)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silica glass (mat3)**.
2. Select Domain 3 only.
3. On the **Model** toolbar, click **Add Material** to close the **Add Material** window.

Material 4 (mat4)

1. In the **Model Builder** window, right-click **Materials** and choose **Blank Material**.
2. In the **Settings** window for Material, type `Thermal Grease` in the **Label** text field.
3. Locate the **Geometric Entity Selection** section. From the **Geometric entity level** list, choose **Boundary**.
4. Select Boundary 34 only.
5. Click to expand the **Material properties** section. Locate the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Thermal Conductivity**.
6. Click **Add to Material**.
7. Locate the **Material Contents** section. In the table, enter the following settings:

Property	Name	Value	Unit	Property group
Thermal conductivity	k	`2[W/m/K]`	W/(m·K)	Basic

DEFINITIONS

Parameters

1. In the **Model Builder** window, expand the **Global>Definitions** node, then click **Parameters**.
2. In the **Settings** window for Parameters, locate the **Parameters** section.
3. In the table, enter the following settings:

Name	Expression	Value	Description
`U0`	`5[cm/s]`	0.05000 m/s	`Mean inlet velocity`
`T0`	`20[degC]`	293.2 K	`Inlet temperature`
`P_tot`	`1[W]`	1.000 W	`Total power dissipated by the electronics package`

Now define the physical properties of the model. Start with the fluid domain.

LAMINAR FLOW (SPF)

On the Physics toolbar, click **Heat Transfer (ht)** and choose **Laminar Flow (spf)**.

The no-slip condition is the default boundary condition for the fluid. Define the inlet and outlet conditions as described below.

Inlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Inlet**.

2 Select Boundary 121 only.

3 In the **Settings** window for Inlet, locate the **Boundary Condition** section.

4 From the list, choose **Laminar inflow**.

5 Locate the **Laminar Inflow** section. In the U_{av} text field, type `U0`.

Outlet 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outlet**.

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

3 Select Boundary 1 only.

HEAT TRANSFER (HT)

On the Physics toolbar, click **Laminar Flow (spf)** and choose **Heat Transfer (ht)**.

Thermal insulation is the default boundary condition for the temperature. Define the inlet temperature and the outlet condition as described below.

Temperature 1

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

2 Select Boundary 121 only.

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

4 In the T_0 text field, type `T0`.

Outflow 1

1 On the **Physics** toolbar, click **Boundaries** and choose **Outflow**.

2 Select Boundary 1 only.

Next, use the `P_tot` parameter to define the total heat source in the electronics package.

Heat Source 1

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

2 Select Domain 3 only.

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

4 Click the **Overall heat transfer rate** button.

5 In the P_{tot} text field, type `P_tot`.

Finally, add the thin thermal grease layer.

Thin Layer 1

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

2 Select Boundary 34 only.

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

4 In the d_s text field, type `50[um]`.

MESH 1

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

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

3 From the **Element size** list, choose **Extra coarse**.

4 Click the **Build All** button.

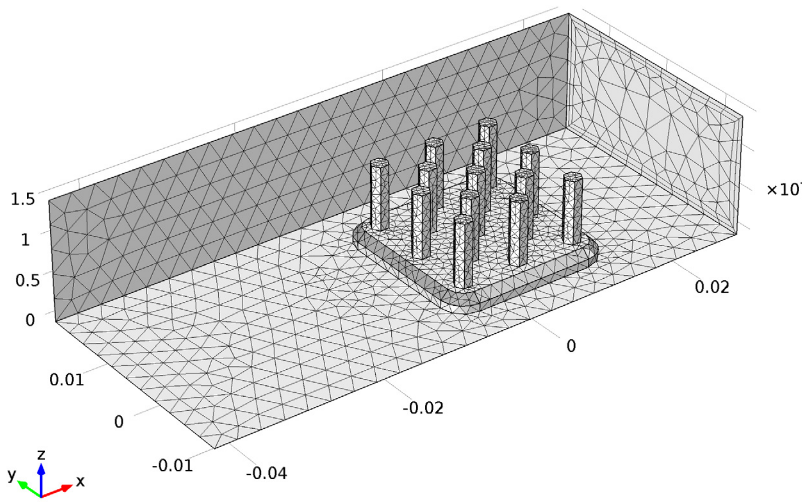
To get a better view of the mesh, hide some of the boundaries.

5 Click the **Select and Hide** button on the **Graphics** toolbar.

6 Click the **Select Boundaries** button on the **Graphics** toolbar.

7 Select Boundaries 1, 2, and 4 only.

The finished mesh should look like that in the figure below.



To achieve more accurate numerical results, this mesh can be refined by choosing another predefined element size. However, doing so requires more computational time and memory.

STUDY 1

On the **Model** toolbar, click **Compute**.

RESULTS

Temperature (ht)

Four default plots are generated automatically. The first one shows the temperature on the wall boundaries, the third one shows the velocity magnitude on five parallel slices, the last one shows the pressure field and. Add an arrow plot to visualize the velocity field with temperature field.]

1 In the **Model Builder** window, under **Results** right-click **Temperature (ht)** and choose **Arrow Volume**.

2 In the **Settings** window for Arrow Volume, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Laminar Flow>u,v,w - Velocity field**.
3 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** text field, type `40`.
4 Find the **y grid points** subsection. In the **Points** text field, type `20`.
5 Find the **z grid points** subsection. From the **Entry method** list, choose **Coordinates**.
6 In the **Coordinates** text field, type `5[mm]`.
7 Right-click **Results>Temperature (ht)>Arrow Volume 1** and choose **Color Expression**.
8 In the **Settings** window for Color Expression, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Laminar Flow>spf.U - Velocity magnitude**.
9 On the **3D plot group** toolbar, click **Plot**.

Derived Values

1 On the **Results** toolbar, click **Global Evaluation**.
2 In the **Settings** window for Global Evaluation, type `Net Energy Rate` in the **Label** text field.
3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer>Global>Net powers>ht.ntefluxInt - Total net energy rate**.
4 Click the **Evaluate** button.

TABLE

Go to the **Table** window.

RESULTS

Derived Values

1 On the **Results** toolbar, click **Global Evaluation**.
2 In the **Settings** window for Global Evaluation, type `Heat Source` in the **Label** text field.
3 Click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Component 1>Heat Transfer>Global>Heat source powers>ht.QInt - Total heat source**.
4 Click the **Evaluate** button.

TABLE

1 Go to the **Table** window.

   The total rate of net energy and heat generation should be close to 1 W.

Appendix: Geometry Modeling Instructions

ROOT

On the **Model** toolbar, click **Add Component** and choose **3D**.

DEFINITIONS

First define the geometry parameters.

Parameters

1 On the **Model** toolbar, click **Parameters**.

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

3 In the table, enter the following settings:

Name	Expression	Value	Description
L_channel	7[cm]	0.07000 m	Channel length
W_channel	3[cm]	0.03000 m	Channel width
H_channel	1.5[cm]	0.01500 m	Channel height
L_chip	1.5[cm]	0.01500 m	Chip size
H_chip	2[mm]	0.002000 m	Chip height

GEOMETRY 1

Build the geometry in three steps. First, import the heat sink geometry from a file.

Import 1 (imp1)

1 On the **Model** toolbar, click **Import**.

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

3 Click **Browse**.

4 Browse to the model's Model Library folder and double-click the file `heat_sink.mphbin`.

5 Click **Import**.

   Next, define a work plane containing the bottom surface of the heat sink to draw the imprints of the chip and of the air channel.

6 Click the **Wireframe Rendering** button on the **Graphics** toolbar.

Work Plane 1 (wp1)

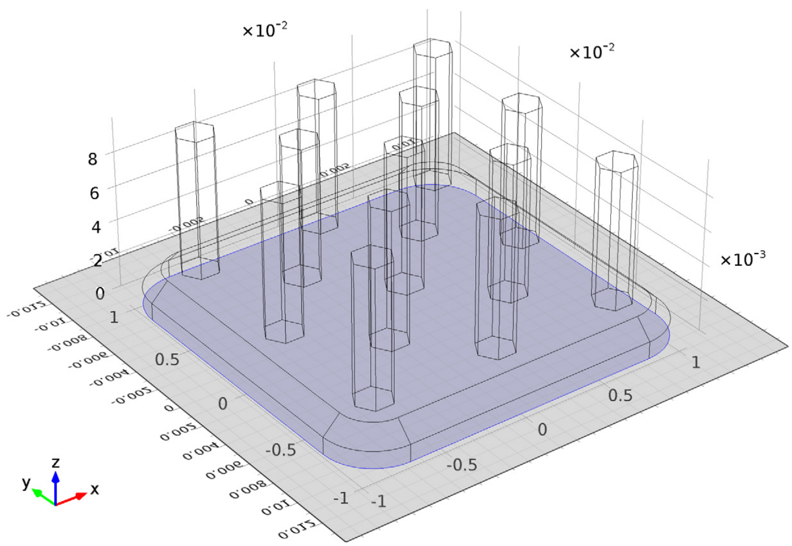
1 On the **Geometry** toolbar, click **Work Plane**.

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

3 From the **Plane type** list, choose **Face parallel**.

4 Find the **Planar face** subsection. Select the **Active** toggle button.

5 Select the surface shown in the figure below.



6 Locate the **Unite Objects** section. Clear the **Unite objects** check box.

Square 1 (sq1)

1 On the **Work plane** toolbar, click **Primitives** and choose **Square**.

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

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

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

5 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Square 1 (sq1)** and choose **Build Selected**.

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

Rectangle 1 (r1)

1 On the **Work plane** toolbar, click **Primitives** and choose **Rectangle**.
2 In the **Settings** window for Rectangle, locate the **Size** section.
3 In the **Width** text field, type `L_channel`.
4 In the **Height** text field, type `W_channel`.
5 Locate the **Position** section. In the **xw** text field, type `-45[mm]`.
6 In the **yw** text field, type `-W_channel/2`.
7 Right-click **Component 1 (comp1)>Geometry 1>Work Plane 1 (wp1)>Plane Geometry>Rectangle 1 (r1)** and choose **Build Selected**.

Now extrude the chip imprint to define the chip volume.

Extrude 1 (ext1)

1 On the **Geometry** toolbar, click **Extrude**.
2 Select the object **wp1.sq1** only.
3 In the **Settings** window for Extrude, locate the **Distances from Plane** section.
4 In the table, enter the following settings:

Distances (m)
H_chip

5 Right-click **Component 1 (comp1)>Geometry 1>Extrude 1 (ext1)** and choose **Build Selected**.
6 Click the **Zoom Extents** button on the **Graphics** toolbar.

To finish the geometry, extrude the channel imprint in the opposite direction to define the air volume.

Extrude 2 (ext2)

1 On the **Geometry** toolbar, click **Extrude**.
2 Select the object **wp1.r1** only.
3 In the **Settings** window for Extrude, locate the **Distances from Plane** section.
4 In the table, enter the following settings:

Distances (m)
H_channel

5 Select the **Reverse direction** check box.
6 Click the **Build All Objects** button.

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

The model geometry is now complete.