Created in COMSOL Multiphysics 5.4

Forced Convection Cooling of an Enclosure with Fan and Grille

Introduction

This study simulates the thermal behavior of a computer power supply unit (PSU). Such electronic enclosures typically include cooling devices to avoid electronic components being damaged by excessively high temperatures. In this application, an extracting fan and a perforated grille generate an airflow in the enclosure to cool internal heating.

Air extracted from the enclosure is related to the static pressure (the pressure difference between outside and inside), information that is generally provided by the fan manufacturers as a curve representing fluid velocity as a function of pressure difference.

As shown in Figure 1, the geometry is rather complicated and requires a fine mesh to solve. This results in large computational costs in terms of time and memory.

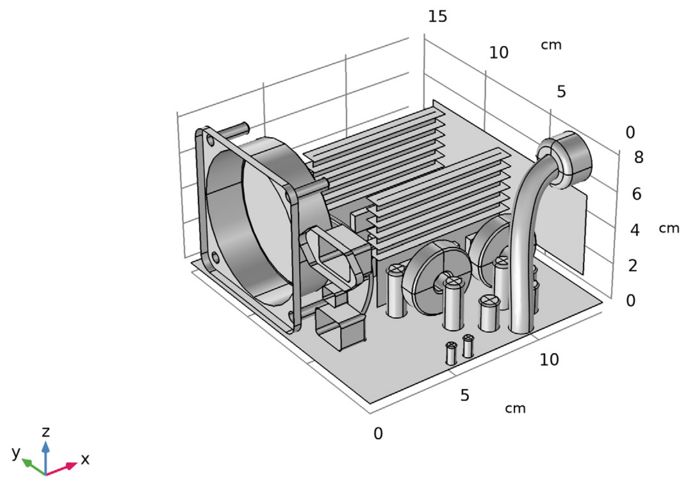


Figure 1: The complete model geometry.

Model Definition

Figure 1 shows the geometry of the PSU. It is composed of a perforated enclosure of 14 cm-by-15 cm-by-8.6 cm and is made of aluminum 6063-T83. Inside the enclosure, only obstacles having a characteristic length of at least 5 mm are represented.

The bottom of the box represents the printed circuit board (PCB). It has an anisotropic thermal conductivity of 10, 10, and 0.36 W/(m·K) along the x-, y-, and z-axes, respectively. Its density is 430 kg/m^3 and its heat capacity at constant pressure is 1100 J/(kg·K). Because the thermal conductivity along the z-axis is relatively low, and that the PCB and the enclosure sides are separated by a thin air layer, it is not necessary to model the bottom wall, nor take into account the cooling on these sides.

The capacitors are approximated by aluminum components. The heat sink fins and the enclosure are made of the same aluminum alloy. The inductors are mainly composed of steel cores and copper coils. The transformers are made of three materials: copper, steel, and plastic. The transistors are modeled as two-domain components: a core made of silicon held in a plastic case. The core is in contact with an aluminum heat sink to allow a more efficient heat transfer. The air flow is be considered as turbulent and is modeled using the Algebraic yPlus turbulence model.

The simulated PSU consumes a maximum of 230 W. Components have been grouped and assigned to various heat sources as listed in Table 1. The overall heat loss is 41 W, which is about 82% of efficiency.

TABLE 1: HEAT SOURCES OF ELECTRONIC COMPONENTS

Components	**Dissipated heat rate (W)**
Transistor cores	25
Large transformer coil	5
Small transformer coils	3
Inductors	2
Large capacitors	2
Medium capacitors	3
Small capacitors	1

The inlet air temperature is set at 30 °C because it is supposed to come from the computer case in which air has already cooled other components. The inlet boundary is configured with a **Grille** boundary condition. This pressure must describe head loss caused by air entry into the enclosure. The head loss coefficient k_{grille} is represented by the following 6^{th} order polynomial (Ref. 1):

$$\begin{aligned} k_{\mathrm{grille}} = {} & 12084\alpha^6 - 42281\alpha^5 + 60989\alpha^4 - 46559\alpha^3 \\ & + 19963\alpha^2 - 4618.5\alpha + 462.89 \end{aligned} \quad (1)$$

where α is the opening ratio of the grille.

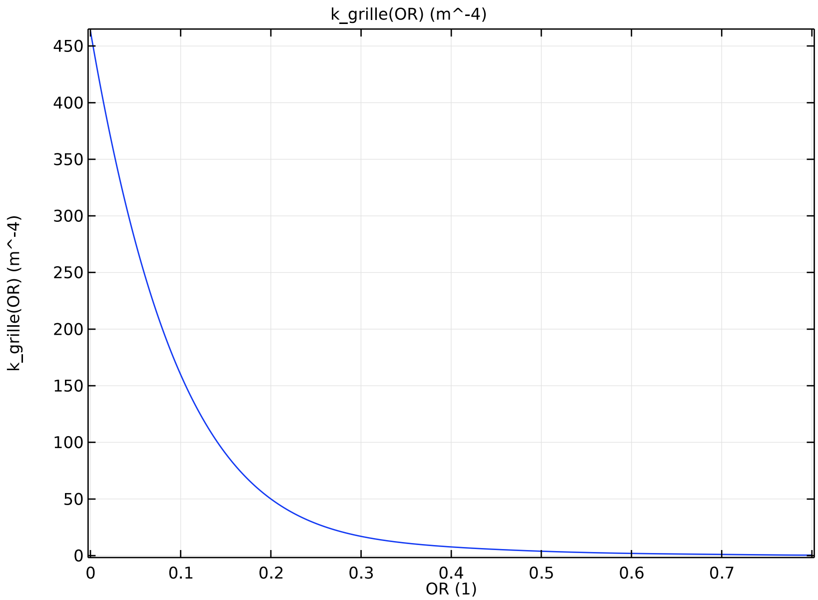


Figure 2: Head loss coefficient as a function of the opening ratio.

The head loss, ΔP (Pa), is given by

$$\Delta P = k_{\text{grille}} \frac{\rho U^2}{2}$$

where ρ is the density (kg/m^3), and U is the velocity magnitude (m/s).

The box, the PCB, the inductor surfaces, and the heat sink fins are configured as thin conductive layers.

Results and Discussion

The most interesting aspect of this simulation is to locate which components are subject to overheating. Figure 3 clearly shows that the temperature distribution is not homogeneous.

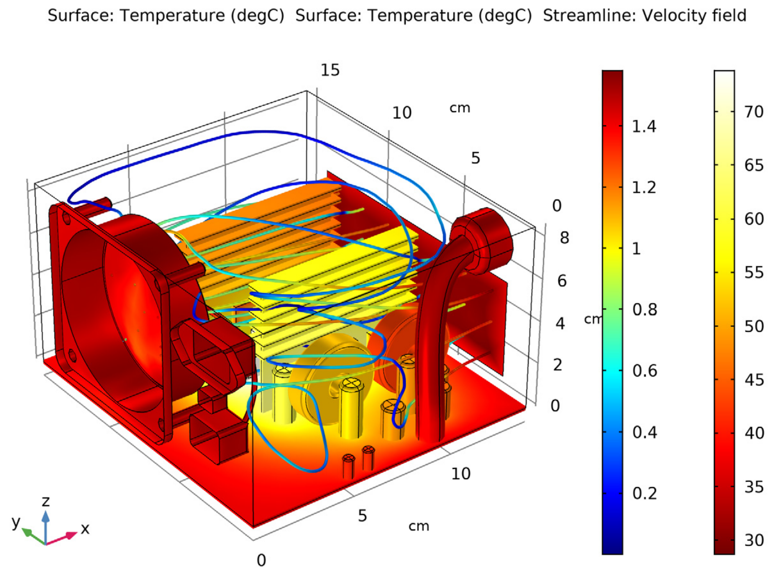


Figure 3: Temperature and fluid velocity fields.

The maximum temperature is about 70 °C and is located at one of the transistor cores. The components furthest away from the air inlet are subject to the highest temperature. Although transistor cores are rather hot, Figure 3 shows that they are significantly cooled by the aluminum heat sinks. The printed circuit board has a significant impact as well by distributing and draining heat off.

On the flow side, air avoids obstacles and tends to go through the upper space of the enclosure. The maximum velocity is about 1.6 m/s.

[Figure 4](#) shows the head loss created by obstacles encountered by air on its path.

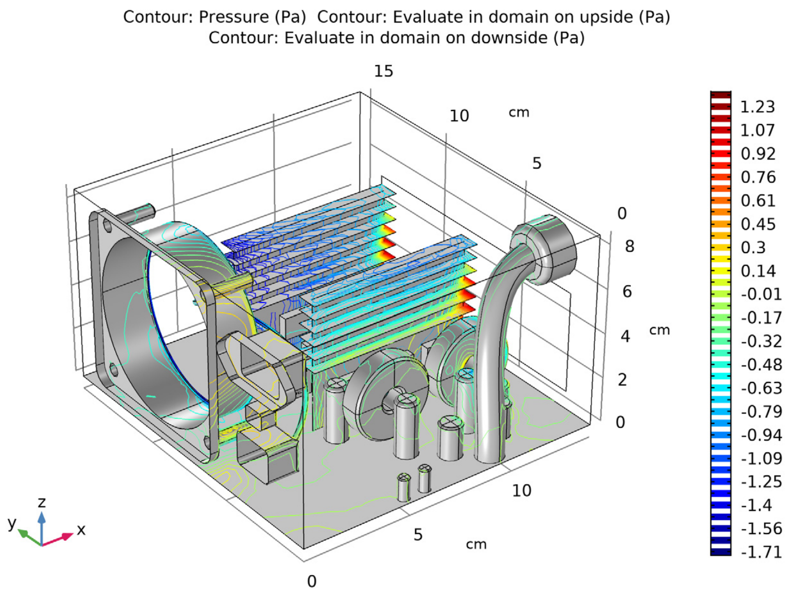


Figure 4: Pressure isosurfaces show the head loss created by electronic components.

As shown in Figure 4, the fluid flow is impacted by obstacles and yields to local head losses.

Notes About the COMSOL Implementation

To model heat sink fins, enclosure walls and circuit board it is strongly recommended to use the **Thin Layer** feature, which is completely adapted for thin geometries and significantly reduces the number of degrees of freedom in the model.

COMSOL Multiphysics provides a useful boundary condition for modeling fan behavior. You just need to provide a few points of a static pressure curve to configure this boundary condition. These data are, most of the time, provided by fan manufacturers.

Reference

1. R.D. Blevins, *Applied Fluid Dynamics Handbook*, Van Nostrand Reinhold, 1984.

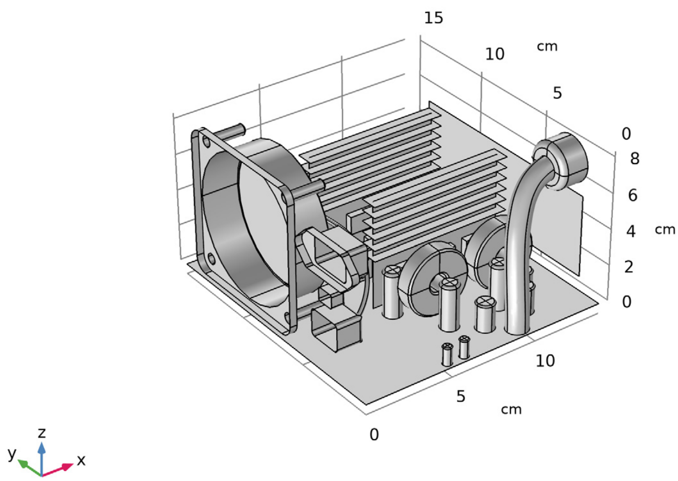
Application Library path: `Heat_Transfer_Module/`
`Power_Electronics_and_Electronic_Cooling/electronic_enclosure_cooling`

Modeling Instructions

ROOT

The file `electronic_enclosure_cooling_geom.mph` contains a parameterized geometry and prepared selections for the model. Start by loading this file.

1. From the **File** menu, choose **Open**.
2. Browse to the model's Application Libraries folder and double-click the file `electronic_enclosure_cooling_geom.mph`.



GLOBAL DEFINITIONS

Define an analytic function to represent the polynomial expression of the head loss coefficient (Equation 1). This coefficient is a function of the open ratio of the grille.

Analytic 1 (an1)

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

2 In the **Settings** window for **Analytic**, type k_grille in the **Function name** text field.

3 Locate the **Definition** section. In the **Expression** text field, type 12084*OR^6-42281*OR^5+60989*OR^4-46559*OR^3+19963*OR^2-4618.5*OR+462.89.

4 In the **Arguments** text field, type OR.

5 Locate the **Units** section. In the **Arguments** text field, type 1.

6 In the **Function** text field, type m^-4.

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

Argument	Lower limit	Upper limit
OR	0	0.8

8 Click **Plot**.

MATERIALS

In this section, you define the materials of the enclosure and its components. The prepared selections make it easier to select the appropriate domains and boundaries.

ADD MATERIAL

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

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

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

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

5 In the tree, select **Built-In>Acrylic plastic**.

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

7 In the tree, select **Built-In>Steel AISI 4340**.

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

9 In the tree, select **Built-In>Aluminum**.

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

11 In the tree, select **Built-In>Copper**.

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

13 In the tree, select **Built-In>Silicon**.

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

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

MATERIALS

Air (mat1)

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

Acrylic plastic (mat2)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Acrylic plastic (mat2)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Plastic**.

Steel AISI 4340 (mat3)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Steel AISI 4340 (mat3)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Steel Parts**.

Aluminum (mat4)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Aluminum (mat4)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Capacitors**.

Copper (mat5)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Copper (mat5)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Transformer Coils**.

Silicon (mat6)

1. In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Silicon (mat6)**.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. From the **Selection** list, choose **Transistors Silicon Cores**.

Single Layer Material 1 (slmat1)

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

2 In the **Settings** window for **Single Layer Material**, type `Heat Sink` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Aluminum Boundaries**.

4 Locate the **Layer Definition** section. In the **Thickness** text field, type `2[mm]`.

5 Click **Add Material from Library**.

ADD MATERIAL TO HEAT SINK (SLMAT1)

1 Go to the **Add Material to Heat Sink (slmat1)** window.

2 In the tree, select **Built-In>Aluminum 6063-T83**.

3 Click **OK**.

MATERIALS

Single Layer Material 2 (slmat2)

1 Right-click **Materials** and choose **Layers>Single Layer Material**.

2 In the **Settings** window for **Single Layer Material**, type `Copper layers` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Copper Layers**.

4 Locate the **Layer Definition** section. In the **Thickness** text field, type `2[mm]`.

5 Click **Add Material from Library**.

ADD MATERIAL TO COPPER LAYERS (SLMAT2)

1 Go to the **Add Material to Copper layers (slmat2)** window.

2 In the tree, select **Built-In>Copper**.

3 Click **OK**.

MATERIALS

Single Layer Material 3 (slmat3)

1 Right-click **Materials** and choose **Layers>Single Layer Material**.

2 In the **Settings** window for **Single Layer Material**, type `Circuit Board` in the **Label** text field.

3 Locate the **Boundary Selection** section. From the **Selection** list, choose **Circuit Board**.

4 Locate the **Layer Definition** section. In the **Thickness** text field, type `2[mm]`.

5 Click **Blank Material**.

Circuit Board (slmat3)

1 In the **Model Builder** window, under **Component 1 (comp1)>Materials** click **Circuit Board (slmat3)**.
2 In the **Settings** window for **Single Layer Material**, locate the **Layer Definition** section.
3 Click **Go to Material**.

GLOBAL DEFINITIONS

Material 9 (mat9)

1 In the **Model Builder** window, under **Global Definitions>Materials** click **Material 9 (mat9)**.
2 In the **Settings** window for **Material**, type `FR4 (Circuit Board)` in the **Label** text field.
3 Click to expand the **Material Properties** section. In the **Material properties** tree, select **Basic Properties>Heat Capacity at Constant Pressure**.
4 Click **Add to Material**.
5 In the **Material properties** tree, select **Basic Properties>Density**.
6 Click **Add to Material**.
7 In the **Material properties** tree, select **Basic Properties>Thermal Conductivity**.
8 Click **Add to Material**.
9 Locate the **Material Contents** section. In the table, enter the following settings:

Property	Variable	Value	Unit	Property group
Heat capacity at constant pressure	Cp	1369[J/(kg*K)]	J/(kg·K)	Basic
Density	rho	1900[kg/m^3]	kg/m³	Basic
Thermal conductivity	k_iso ; kii = k_iso, kij = 0		W/(m·K)	Basic

10 Right-click the **Thermal conductivity** row and select **Edit**
11 In the **Thermal conductivity** dialog box, choose **Diagonal**.
12 In the **Diagonal** table, enter the following settings:

10	0	0
0	10	0
0	0	0.3

13 click **OK**.

The next steps define the boundary conditions of the model.

DEFINITIONS (COMP1)

Ambient Thermal Properties 1 (amth1)

1 In the **Physics** toolbar, click **Ambient Thermal Properties**.

Set the ambient temperature to be used in initial values of the Heat Transfer interface.

2 In the **Settings** window for **Ambient Thermal Properties**, locate the **Ambient Conditions** section.

3 In the T_{amb} text field, type `30[degC]`.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

Fluid 1

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

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

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

Initial Values 1

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

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

3 From the T list, choose **Ambient temperature (amth1)**.

Now, define the heat rate produced by the different components. Table 1 summarizes the values chosen for each kind of component.

Heat Source 1

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

2 In the **Settings** window for **Heat Source**, type `Heat Source 1: Transistors` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Transistors Silicon Cores**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `25`.

Heat Source 2

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 In the **Settings** window for **Heat Source**, type `Heat Source 2: Large Transformer Coil` in the **Label** text field.
3 Locate the **Domain Selection** section. From the **Selection** list, choose **Large Transformer Coil**.
4 Locate the **Heat Source** section. Click the **Heat rate** button.
5 In the P_0 text field, type `5`.

Heat Source 3

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 In the **Settings** window for **Heat Source**, type `Heat Source 3: Small Transformer Coils` in the **Label** text field.
3 Locate the **Heat Source** section. Click the **Heat rate** button.
4 In the P_0 text field, type `3`.
5 Locate the **Domain Selection** section. From the **Selection** list, choose **Small Transformer Coils**.

Heat Source 4

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 In the **Settings** window for **Heat Source**, type `Heat Source 4: Inductor` in the **Label** text field.
3 Locate the **Domain Selection** section. From the **Selection** list, choose **Inductors**.
4 Locate the **Heat Source** section. Click the **Heat rate** button.
5 In the P_0 text field, type `2`.

Heat Source 5

1 In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2 In the **Settings** window for **Heat Source**, type `Heat Source 5: Large Capacitors` in the **Label** text field.
3 Locate the **Domain Selection** section. From the **Selection** list, choose **Large Capacitors**.
4 Locate the **Heat Source** section. Click the **Heat rate** button.
5 In the P_0 text field, type `2`.

Heat Source 6

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

2 In the **Settings** window for **Heat Source**, type `Heat Source 6: Medium Capacitors` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Medium Capacitors**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `3`.

Heat Source 7

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

2 In the **Settings** window for **Heat Source**, type `Heat Source 7: Small Capacitors` in the **Label** text field.

3 Locate the **Domain Selection** section. From the **Selection** list, choose **Small Capacitors**.

4 Locate the **Heat Source** section. Click the **Heat rate** button.

5 In the P_0 text field, type `1`.

Inflow 1

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

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

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

4 Locate the **Upstream Properties** section. From the T_{ustr} list, choose **Ambient temperature (amth1)**.

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 **Conductive Layers**.

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

Outflow 1

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

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

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

TURBULENT FLOW, ALGEBRAIC YPLUS (SPF)

1 In the **Model Builder** window, under **Component 1 (comp1)** click **Turbulent Flow, Algebraic yPlus (spf)**.

2 In the **Settings** window for **Turbulent Flow, Algebraic yPlus**, locate the **Domain Selection** section.

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

4 Locate the **Turbulence** section. From the **Wall treatment** list, choose **Low Re**.

The thin heat sink fins are represented by interior boundaries in the geometry. An interior wall condition is used to prevent the fluid from flowing through these boundaries.

Interior Wall 1

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

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

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

Fan 1

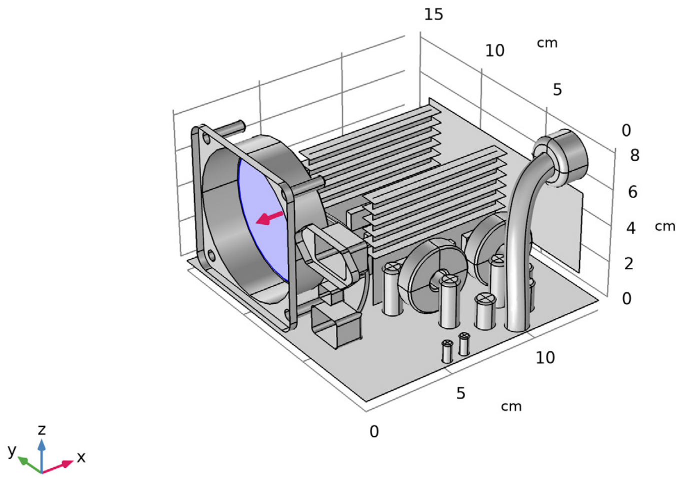
1 In the **Physics** toolbar, click **Boundaries** and choose **Fan**.

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

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

4 Locate the **Flow Direction** section. From the **Flow direction** list, choose **Outlet**.

The **Graphics** window displays an arrow indicating the orientation of the flow through the fan. Compare with the figure below.



Here, the **Fan** condition is set up by loading a data file for the static pressure curve.

5 Locate the **Parameters** section. From the **Static pressure curve** list, choose **Static pressure curve data**.

6 Locate the **Static Pressure Curve Data** section. Click **Load from File**.

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

8 Locate the **Static Pressure Curve Interpolation** section. From the **Interpolation function type** list, choose **Piecewise cubic**.

The exhaust fan previously defined extracts air entering from an opposite grille. Proceed to create the corresponding boundary condition.

Grille 1

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

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

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

4 Locate the **Parameters** section. In the *qlc* text field, type `k_grille(OR)*nitf1.rho/2`.

Initial Values 1

1 In the **Model Builder** window, under **Component 1 (comp1)>Turbulent Flow, Algebraic yPlus (spf)** click **Initial Values 1**.

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

3 Specify the **u** vector as

-1	x

MESH 1

You now configure the meshing part. Start by discretizing the surfaces of key components. They would drive the tetrahedral mesh of the whole domain. Boundary layers at walls are added at the end.

Size

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.

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

3 From the **Predefined** list, choose **Coarse**.

Mapped 1

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

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

3 From the **Selection** list, choose **Wire Group Surface**.

4 Click to expand the **Advanced Settings** section. In order to get a regular mesh, select the option to adjust the mesh position on the edges in the swept direction.

5 Select the **Adjust evenly distributed edge mesh** check box.

Size 1

1 Right-click **Component 1 (comp1)>Mesh 1>Mapped 1** and choose **Size**.

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

3 From the **Predefined** list, choose **Finer**.

4 Click the **Custom** button.

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

6 In the associated text field, type `0.5`.

7 Select the **Minimum element size** check box.

8 In the associated text field, type `0.4`.

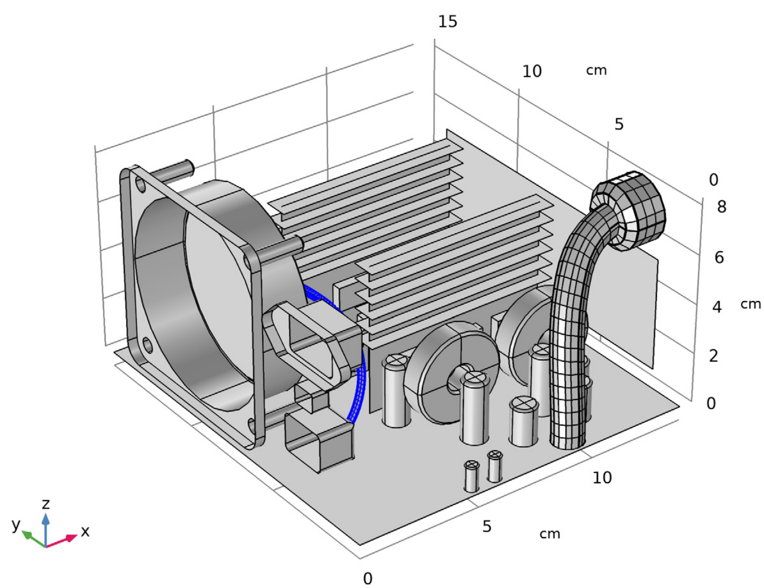
9 Click **Build Selected**.

Mapped 2

1 In the **Mesh** toolbar, click **Boundary** and choose **Mapped**.
2 In the **Settings** window for **Mapped**, locate the **Boundary Selection** section.
3 From the **Selection** list, choose **Small Wire Surface**.
4 Locate the **Advanced Settings** section. Select the **Adjust evenly distributed edge mesh** check box.

Size 1

1 Right-click **Mapped 2** and choose **Size**.
2 In the **Settings** window for **Size**, locate the **Element Size** section.
3 From the **Calibrate for** list, choose **Fluid dynamics**.
4 From the **Predefined** list, choose **Extra fine**.
5 Click the **Custom** button.
6 Locate the **Element Size Parameters** section. Select the **Maximum element size** check box.
7 In the associated text field, type `0.16`.
8 Click **Build Selected**.



Free Triangular 1

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

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

3 From the **Selection** list, choose **Heat Exchange Surface**.

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. Select the **Maximum element size** check box.

5 In the associated text field, type `0.5`.

6 Select the **Minimum element size** check box.

7 In the associated text field, type `0.4`.

8 Select the **Maximum element growth rate** check box.

9 In the associated text field, type `1.05`.

10 Select the **Curvature factor** check box.

11 In the associated text field, type `1`.

12 Select the **Resolution of narrow regions** check box.

13 In the associated text field, type `1`.

14 Click **Build Selected**.

Free Triangular 2

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

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

3 From the **Selection** list, choose **Curved Area**.

Size 1

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

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

3 From the **Calibrate for** list, choose **Fluid dynamics**.

4 Click **Build Selected**.

Boundary Layers 1

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

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

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

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

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

6 Click to expand the **Corner Settings** section. From the **Handling of sharp edges** list, choose **Trimming**.

Boundary Layer Properties 1

1 In the **Model Builder** window, expand the **Boundary Layers 1** node, then click **Boundary Layer Properties 1**.

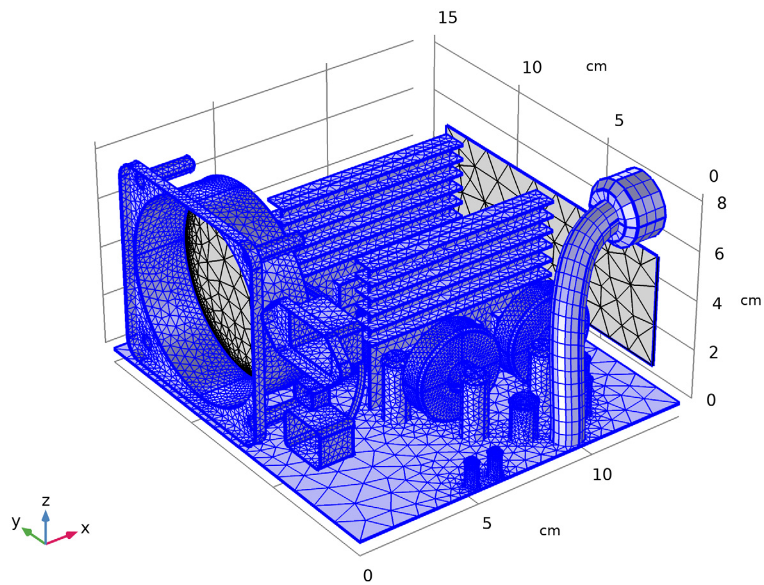
2 In the **Settings** window for **Boundary Layer Properties**, locate the **Boundary Selection** section.

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

4 Locate the **Boundary Layer Properties** section. In the **Number of boundary layers** text field, type 3.

5 In the **Thickness adjustment factor** text field, type 5.

6 Click **Build All**.



STUDY 1

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

RESULTS

Temperature (ht)

The first default plot shows the temperature field. To reproduce the plot in Figure 3 of the temperature and the air velocity, proceed as follows.

Surface 1

1 In the **Model Builder** window, expand the **Temperature (ht)** node, then click **Surface 1**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **degC**.

Surface 2

1 In the **Model Builder** window, under **Results>Temperature (ht)** click **Surface 2**.
2 In the **Settings** window for **Surface**, locate the **Expression** section.
3 From the **Unit** list, choose **degC**.

Temperature (ht)

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

Streamline 1

1 In the **Temperature (ht)** toolbar, click **Streamline**.
2 In the **Settings** window for **Streamline**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1>Turbulent Flow, Algebraic yPlus>Velocity and pressure>u,v,w - Velocity field**.
3 Locate the **Selection** section. From the **Selection** list, choose **Grille**.
4 Locate the **Coloring and Style** section. Find the **Line style** subsection. From the **Type** list, choose **Tube**.

Color Expression 1

1 In the **Temperature (ht)** toolbar, click **Color Expression**.
2 In the **Settings** window for **Color Expression**, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Model>Component 1> Turbulent Flow, Algebraic yPlus>Velocity and pressure>spf.U - Velocity magnitude - m/s**.
3 In the **Temperature (ht)** toolbar, click **Plot**.

Velocity (spf)

The third default plot group shows the air velocity profile in a slice plot.

Pressure (spf)

This last default plot group shows the pressure field plot as in Figure 4.