Model created in COMSOL Multiphysics 6.4

Convection Cooling of Circuit Boards — 2D Natural Convection

Introduction

This example models the air cooling of circuit boards populated with multiple integrated circuits (ICs), which act as heat sources. Two possible cooling scenarios are shown in Figure 1: vertically aligned boards using natural convection, and horizontal boards with forced convection (fan cooling). In this case, contributions caused by the induced (forced) flow of air dominate the cooling. To achieve high accuracy, the simulation models heat transport in combination with the fluid flow.

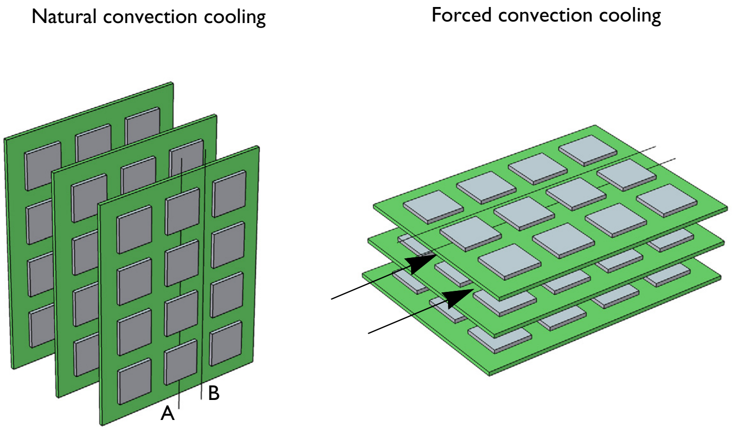

Figure 1: Stacked circuit boards with multiple in-line heat sources. Line A represents the centerline of the row of ICs, and the area between lines A–B on the board represents the symmetry.

A common technique is to describe convective heat flux with a film-resistance coefficient, h. The heat-transfer equations then become simple to solve. However, this simplification requires that the coefficient is well determined which is difficult for many systems and conditions.

An alternative way to thoroughly describe the convective heat transfer is to model the heat transfer in combination with the fluid-flow field. The results then accurately describe the heat transport and temperature changes. From such simulations it is also possible to derive accurate estimations of the film coefficients. Such models are somewhat more complex but they are useful for unusual geometries and complex flows. The following example models the heat transfer of a circuit-board assembly using the Conjugate Heat Transfer predefined multiphysics coupling of the Heat Transfer Module. The modeled scenario is based on work published by A. Ortega (Ref. 1).

FR4 circuit board material (Ref. 2) and silicon are used as the solid materials composing the circuit board system. The model treats air properties as temperature dependent.

The dimensions of the original geometry are:

- Board: length (in the flow direction) 130 mm, and the thickness is 2 mm
- ICs: length and width are both 20 mm, and thickness is 2 mm
- The boards are spaced 10 mm apart.

Model Definition

This example models the natural convection. The simplified model considers the 2D cross section, from the board's back side to the next board's back side, through the center of a row of ICs (as indicated by line A in Figure 1). Figure 2 depicts the 2D geometry.

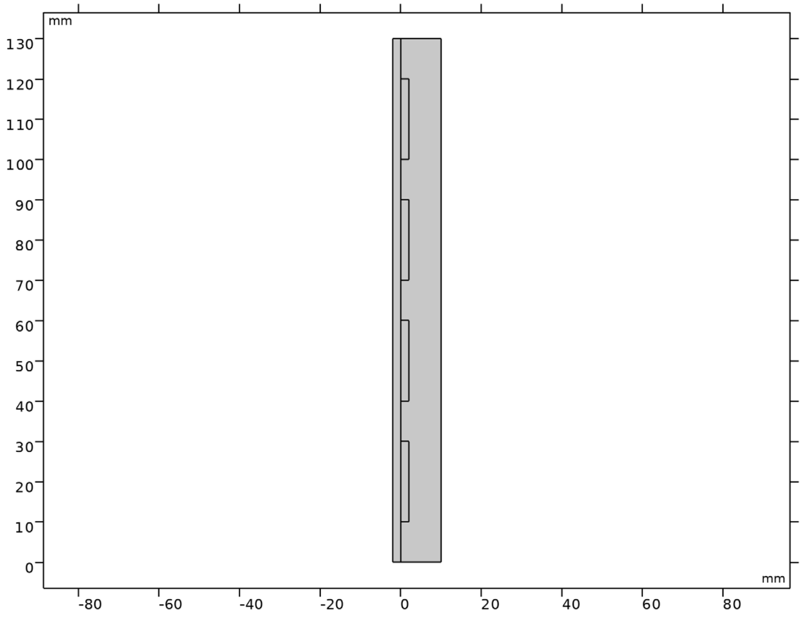

Figure 2: The modeled geometry in 2D.

The model makes use of the Conjugate Heat Transfer predefined multiphysics coupling with a stationary study to set up the simulation. The heating rate per unit volume in the solid is 0.83 MW/m^3. It is 2/3 of the real heat power in order to represent the lateral average heating power (that is, taking into account the open slots between the ICs).

Due to heating of the fluid, deviations occur in the local density, ρ, compared to the inlet density, ρ_0. This results in a local buoyancy force defined using the **Gravity** feature in the **Single Phase Flow** interface.

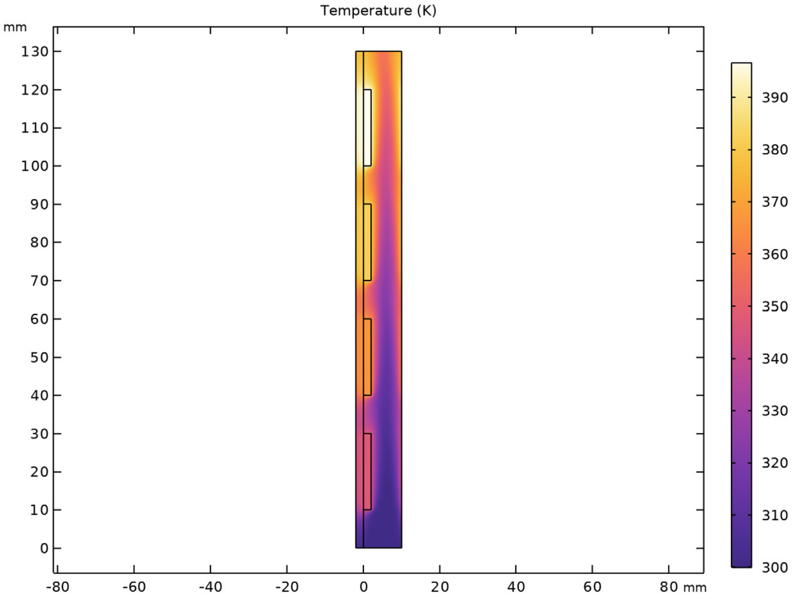

Figure 3: Temperature distribution.

BOUNDARY CONDITIONS

The top and bottom boundaries are open boundaries with given external air temperature. In addition, no slip conditions at the surfaces of the board and the ICs are applied. You should also set the lateral boundaries periodic with respect to temperature, making the temperatures equal on both boundaries at every value. Finally, the models apply continuity of temperature and heat flux at all interior boundaries.

Results and Discussion

The results (Figure 3) show that the temperature of the ICs (the heat sources) increases considerably under a constant heating load from the components. Note that the temperature increase of the sources varies from 50 K for the lowest IC up to 100 K at the top IC. This is a result of the thermal “footprint” of the heat sources. Another interesting result is that the circuit board contributes a large amount of cooling power on its back side, although the thermal conductivity is quite small. This is apparent in the result plots as a temperature rise in the fluid at the right-hand boundary (that is, the back side of the next board in the stack).

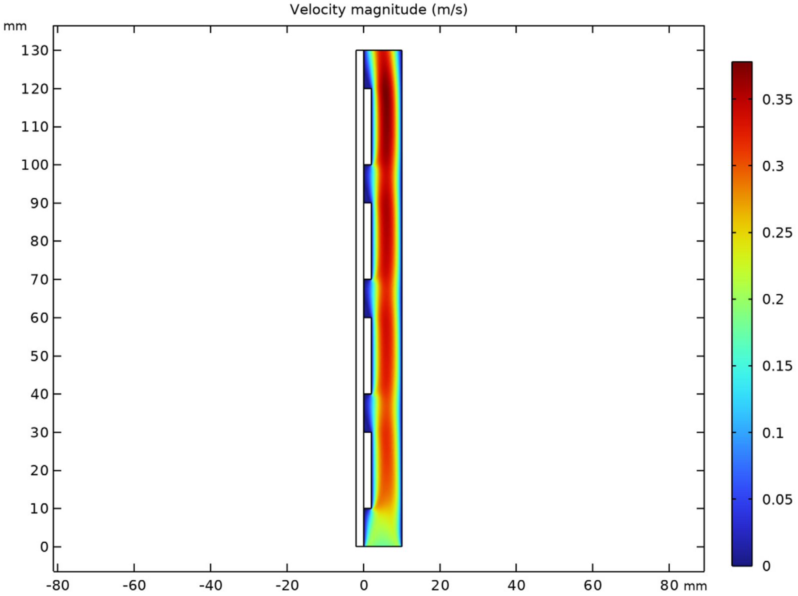

Figure 4: Velocity field.

References

1. A. Ortega, "Air Cooling of Electronics: A Personal Perspective 1981-2001," presentation material, *IEEE SMITHERM* Symposium, 2002.

2. C. Bailey, "Modeling the Effect of Temperature on Product Reliability," Proc. 19th *IEEE SMITHERM* Symposium, 2003.

Application Library path: `Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/circuit_board_nat_2d`

Modeling Instructions

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

NEW

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

MODEL WIZARD

1. In the **Model Wizard** window, click **2D**.
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 **General Studies** > **Stationary**.
6. Click **Done**.

GLOBAL DEFINITIONS

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
`q_source`	`(2/3)*1[W]/(20*20*2[mm^3])`	8.3333E5 W/m^3	Heating power per unit volume
`T0`	`300[K]`	300 K	External air temperature
`patm`	`1[atm]`	1.0133E5 Pa	Air pressure

GEOMETRY 1

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Geometry 1**.
2. In the **Settings** window for **Geometry**, locate the **Units** section.
3. From the **Length unit** list, choose **mm**.

Rectangle 1 (r1)

1. In the **Geometry** toolbar, click **Rectangle**.
2. In the **Settings** window for **Rectangle**, locate the **Size and Shape** section.
3. In the **Width** text field, type `12`.

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

5 Locate the **Position** section. In the **x** text field, type `-2`.

6 Click to expand the **Layers** section. Select the **Layers to the left** checkbox.

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

8 In the table, enter the following settings:

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

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

Rectangle 2 (r2)

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

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

3 In the **Width** text field, type `2`.

4 In the **Height** text field, type `20`.

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

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

Array 1 (arr1)

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

2 Select the object **r2** only.

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

4 In the **y size** text field, type `4`.

5 Locate the **Displacement** section. In the **y** text field, type `30`.

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

The geometry should now look like that in Figure 2.

DEFINITIONS

IC

1 In the **Definitions** toolbar, click **Explicit**.

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

3 Select Domains 3–6 only.

ADD MATERIAL

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

2. Go to the **Add Material** window.
3. In the tree, select **Built-in** > **Air**.
4. Click the **Add to Component** button in the window toolbar.
5. In the tree, select **Built-in** > **Silicon**.
6. Click the **Add to Component** button in the window toolbar.
7. In the tree, select **Built-in** > **FR4 (Circuit Board)**.
8. Click the **Add to Component** button in the window toolbar.
9. In the **Materials** toolbar, click **Add Material** to close the **Add Material** window.

MATERIALS

Air (mat1)

1. Select Domain 2 only.
2. In the **Settings** window for **Material**, locate the **Geometric Entity Selection** section.
3. Click **Create Selection**.
4. In the **Create Selection** dialog, type `Air` in the **Selection name** text field.
5. Click **OK**.

Silicon (mat2)

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

FR4 (Circuit Board) (mat3)

1. In the **Model Builder** window, click **FR4 (Circuit Board) (mat3)**.
2. Select Domain 1 only.

LAMINAR FLOW (SPF)

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. From the **Selection** list, choose **Air**.
4. Locate the **Physical Model** section. Select the **Include gravity** checkbox.

Open Boundary 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.
2. Select Boundaries 5 and 22 only.

HEAT TRANSFER IN SOLIDS AND FLUIDS (HT)

1. In the **Model Builder** window, under **Component 1 (comp1)** click **Heat Transfer in Solids and Fluids (ht)**.
2. In the **Settings** window for **Heat Transfer in Solids and Fluids**, locate the **Physical Model** section.
3. In the T_{ref} text field, type `T0`.

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**.

Open Boundary 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Open Boundary**.
2. Select Boundaries 5 and 22 only.
3. In the **Settings** window for **Open Boundary**, locate the **Upstream Properties** section.
4. In the T_{ustr} text field, type `T0`.

Heat Source 1

1. In the **Physics** toolbar, click **Domains** and choose **Heat Source**.
2. In the **Settings** window for **Heat Source**, locate the **Domain Selection** section.
3. From the **Selection** list, choose **IC**.
4. Locate the **Heat Source** section. In the Q_0 text field, type `q_source`.

Periodic Condition 1

1. In the **Physics** toolbar, click **Boundaries** and choose **Periodic Condition**.
2. Select Boundaries 1 and 27 only.

STUDY 1

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

RESULTS

Temperature (ht)

The first default plot shows the temperature distribution as in Figure 3.

Velocity (spf)

This default plot shows the velocity magnitude as in Figure 4.