Convection Cooling of Circuit Boards—3D 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 depicted 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.

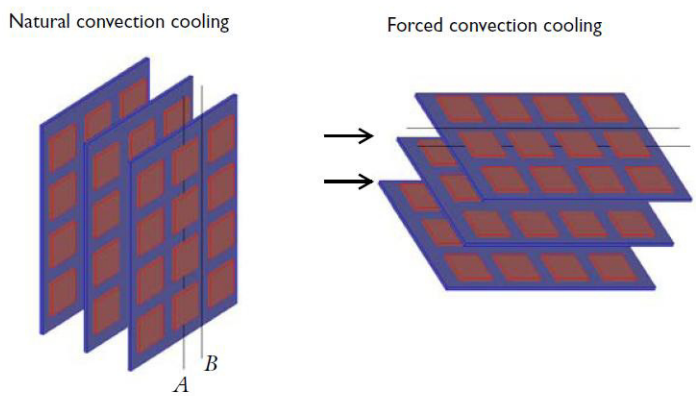

Figure 1: Stacked circuit boards with multiple in-line heat sources. Line A represents the center line of the row of ICs, and the area between lines A-B on the board represents the symmetry.

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 Heat Transfer Module's Conjugate Heat Transfer predefined multiphysics coupling. 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 distance of air between the boards is 10 mm

Model Definition

This example simulates natural convection cooling of a vertical circuit board as depicted in Figure 1. Due to symmetry, it is sufficient to model a unit cell, from the back side of a board to the next back side, covering the area between lines A and B in Figure 1. Figure 2 depicts the three-dimensional geometry.

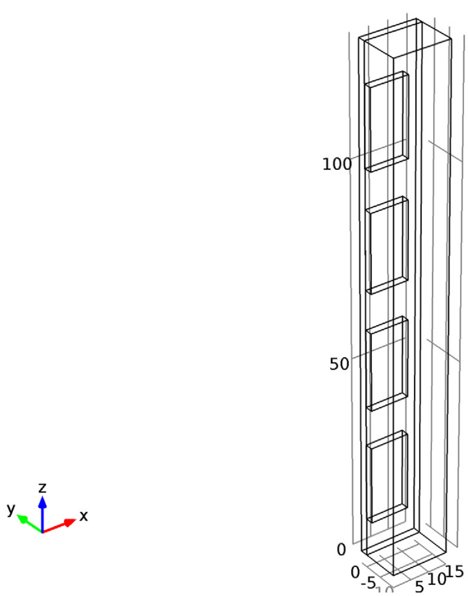

Figure 2: The modeled geometry.

The model makes use of the Conjugate Heat Transfer interface with a stationary study. The heating power per unit volume is 1.25 MW/m^3. 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 expressed as $(\rho - \rho_0)\mathbf{g}$.

Results and Discussion

The temperature distribution is shown on Figure 3. The temperature increase at the hottest spot of each component computed in this 3D model is approximately two degrees higher than that for the 2D model (see Convection Cooling of Circuit Boards—2D Natural Convection for 2D model description and results). In addition, the temperature difference among the various ICs is smaller in the 3D model, which predicts a more uniform temperature rise of the ICs. The ICs have an operating temperature between 70 K and 80 K above ambient. This result is closer to reality compared to the 2D simulation because it also includes the horizontal gaps between the ICs.

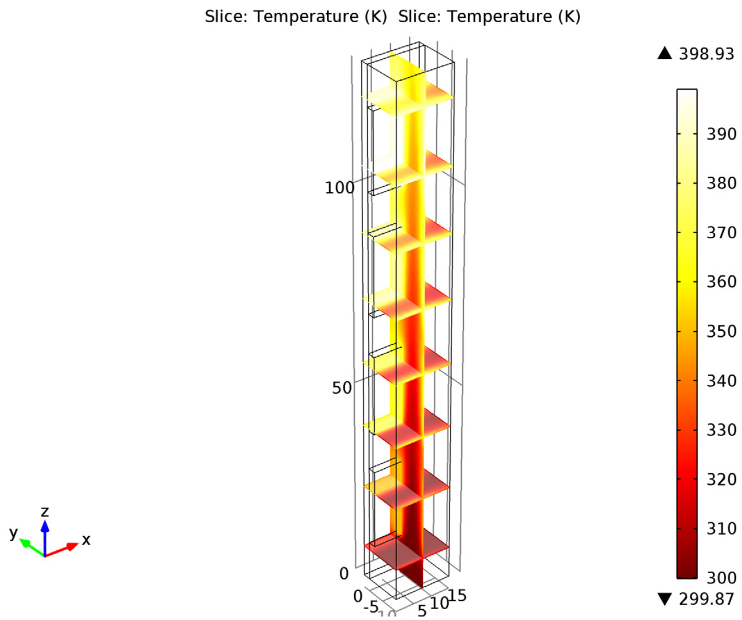

Figure 3: Temperature distribution for 3D model

The difference in temperature rise along the board’s height is explained primarily by the fluid-flow pattern (Figure 4). The maximum fluid velocity is slightly higher for the 3D case than for the 2D case. More importantly, the flow field behaves differently in the 3D case. When making a comparison between the 2D and 3D models, it can be noticed the velocity fields are rather similar along the center line of the heat sources. However, there is a channeling effect from the horizontal gaps.

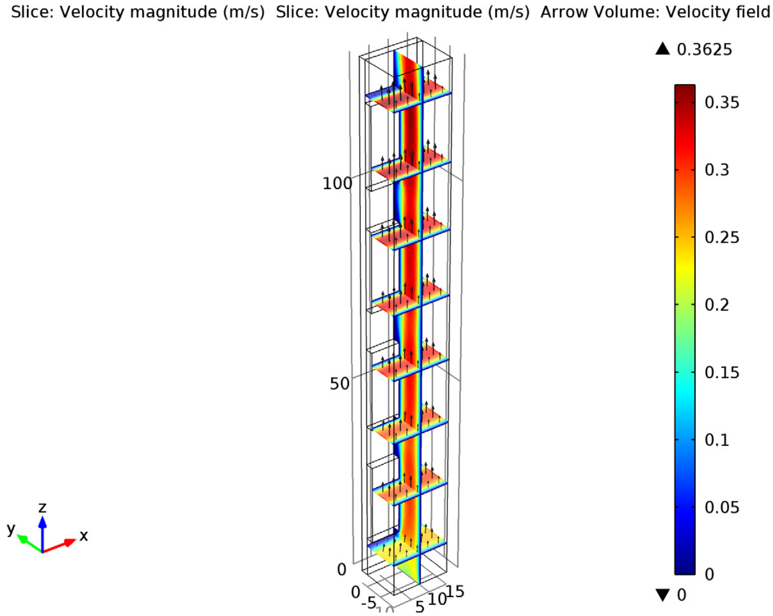

Figure 4: Velocity field distribution.

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.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/circuit_board_nat_3d`

Modeling Instructions

MODEL WIZARD

1 Go to the **Model Wizard** window.

2 Click **Next**.

3 In the **Add physics** tree, select **Heat Transfer>Conjugate Heat Transfer>Laminar Flow (nitf)**.

4 Click **Add Selected**.

5 Click **Next**.

6 Find the **Studies** subsection. In the tree, select **Preset Studies>Stationary**.

7 Click **Finish**.

GLOBAL DEFINITIONS

Parameters

1 In the **Model Builder** window, right-click **Global Definitions** and choose **Parameters**.

2 In the **Parameters** settings window, locate the **Parameters** section.

3 In the table, enter the following settings:

NAME	EXPRESSION	DESCRIPTION
q_source	1[W]/(20*20*2[mm^3])	Heating power per unit volume
T0	300[K]	External air temperature
patm	1[atm]	Air pressure

GEOMETRY 1

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

2 In the **Geometry** settings window, locate the **Units** section.

3 From the **Length unit** list, choose **mm**.

Block 1

1 Right-click **Model 1>Geometry 1** and choose **Block**.

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

3 In the **Width** edit field, type 15.

4 In the **Depth** edit field, type 2.

5 In the **Height** edit field, type 130.

Block 2

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.
2 In the **Block** settings window, locate the **Size and Shape** section.
3 In the **Width** edit field, type 10.
4 In the **Depth** edit field, type 2.
5 In the **Height** edit field, type 20.
6 Locate the **Position** section. In the **y** edit field, type 2.
7 In the **z** edit field, type 10.
8 In the **y** edit field, type -2.

Block 3

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Block**.
2 In the **Block** settings window, locate the **Size and Shape** section.
3 In the **Width** edit field, type 15.
4 In the **Depth** edit field, type 10.
5 In the **Height** edit field, type 130.
6 Locate the **Position** section. In the **y** edit field, type -10.

Array 1

1 In the **Model Builder** window, right-click **Geometry 1** and choose **Transforms>Array**.
2 Select the object **blk2** only.
3 In the **Array** settings window, locate the **Size** section.
4 In the **z size** edit field, type 4.
5 Locate the **Displacement** section. In the **z** edit field, type 30.
6 Click the **Build All** button.
7 Click the **Wireframe Rendering** button on the Graphics toolbar.

DEFINITIONS

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Selections>Explicit**.
2 Select Domains 2–5 only.
3 Right-click **Model 1>Definitions** and choose **Rename**.
4 Go to the **Rename Explicit** dialog box and type IC in the **New name** edit field.
5 Click **OK**.

MATERIALS

1 In the **Model Builder** window, under **Model 1** right-click **Materials** and choose **Open Material Browser**.

2 In the **Material Browser** window, locate the **Materials** section.

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

4 Right-click and choose **Add Material to Model** from the menu.

Air

1 In the **Model Builder** window, under **Model 1>Materials** click **Air**.

2 Select Domain 1 only.

3 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

4 In the **Material Browser** window, locate the **Materials** section.

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

6 Right-click and choose **Add Material to Model** from the menu.

Silicon

1 In the **Model Builder** window, under **Model 1>Materials** click **Silicon**.

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

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

4 In the **Model Builder** window, right-click **Materials** and choose **Open Material Browser**.

5 In the **Material Browser** window, locate the **Materials** section.

6 In the tree, select **Built-In>FR4 (Circuit Board)**.

7 Right-click and choose **Add Material to Model** from the menu.

FR4 (Circuit Board)

1 In the **Model Builder** window, under **Model 1>Materials** click **FR4 (Circuit Board)**.

2 Select Domain 6 only.

DEFINITIONS

Variables 1

1 In the **Model Builder** window, under **Model 1** right-click **Definitions** and choose **Variables**.

2 In the **Variables** settings window, locate the **Variables** section.

3 In the table, enter the following settings:

NAME	EXPRESSION
rho_ref	mat1.def.rho(patm[1/Pa],T0[1/K])[kg/m^3]

This calculates the density for the given pressure and the temperature. Note that the numerical values of the pressure and the temperature are specified as arguments.

CONJUGATE HEAT TRANSFER

Fluid 1

1 In the **Model Builder** window, right-click **Model 1>Conjugate Heat Transfer** and choose the domain setting **Fluid**.

2 Select Domain 1 only.

3 In the **Fluid** settings window, locate the **Model Inputs** section.

4 In the p_A edit field, type `patm`.

Volume Force 1

1 In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the domain setting **Laminar Flow>Volume Force**.

2 Select Domain 1 only.

3 In the **Volume Force** settings window, locate the **Volume Force** section.

4 Specify the **F** vector as

0	x
0	y
-g_const*(nitf.rho-rho_ref)	z

Here, g_const is a predefined constant for the acceleration of gravity.

Heat Source 1

1 In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the domain setting **Heat Transfer in Solids>Heat Source**.

2 In the **Heat Source** settings window, locate the **Domain Selection** section.

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

4 Locate the **Heat Source** section. In the Q edit field, type `q_source`.

Initial Values 1

1. In the **Model Builder** window, under **Model 1>Conjugate Heat Transfer** click **Initial Values 1**.
2. In the **Initial Values** settings window, locate the **Initial Values** section.
3. In the T edit field, type `T0`.

Symmetry 1

1. In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the boundary condition **Laminar Flow>Symmetry**.
2. Select Boundaries 1 and 34 only.

Open Boundary 1

1. In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the boundary condition **Open Boundary**.
2. Select Boundaries 3 and 4 only.
3. In the **Open Boundary** settings window, locate the **Exterior Temperature** section.
4. In the T_0 edit field, type `T0`.

Periodic Heat Condition 1

1. In the **Model Builder** window, right-click **Conjugate Heat Transfer** and choose the boundary condition **Heat Transfer in Solids>Periodic Heat Condition**.
2. Select Boundaries 2 and 29 only.
3. Right-click **Model 1>Conjugate Heat Transfer>Periodic Heat Condition 1** and choose the boundary condition **Destination Selection**.

MESH 1

In the **Model Builder** window, under **Model 1** right-click **Mesh 1** and choose **Build All**.

STUDY 1

In the **Model Builder** window, right-click **Study 1** and choose **Compute**.

RESULTS

Velocity (nitf)

The first default plot group shows the velocity magnitude on slices.

Modify this plot group to reproduce Figure 4.

1. In the **Model Builder** window, expand the **Velocity (nitf)** node, then click **Slice 1**.

2 In the **Slice** settings window, locate the **Plane Data** section.

3 In the **Planes** edit field, type 1.

4 In the **Model Builder** window, right-click **Velocity (nitf)** and choose **Slice**.

5 In the **Slice** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Conjugate Heat Transfer (Laminar Flow)>Velocity magnitude (nitf.U)**.

6 Locate the **Plane Data** section. From the **Plane** list, choose **xy-planes**.

7 In the **Planes** edit field, type 8.

8 Locate the **Coloring and Style** section. Clear the **Color legend** check box.

9 In the **Model Builder** window, right-click **Velocity (nitf)** and choose **Arrow Volume**.

10 In the **Arrow Volume** settings window, click **Replace Expression** in the upper-right corner of the **Expression** section. From the menu, choose **Conjugate Heat Transfer (Laminar Flow)>Velocity field (u,v,w)**.

11 Locate the **Arrow Positioning** section. Find the **x grid points** subsection. In the **Points** edit field, type 5.

12 Find the **y grid points** subsection. In the **Points** edit field, type 5.

13 Find the **z grid points** subsection. In the **Points** edit field, type 8.

14 Locate the **Coloring and Style** section. From the **Color** list, choose **Black**.

15 Click the **Plot** button.

Temperature (nitf)

The second default plot shows the temperature distribution on surfaces.To reproduce Figure 3, modify this plot group as follows:

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

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

3 From the **Data set** list, choose **Solution 1**.

4 In the **Model Builder** window, expand the **Temperature (nitf)** node.

5 Right-click **Surface 1** and choose **Delete**.

Click **Yes** to confirm.

6 Right-click **Temperature (nitf)** and choose **Slice**.

7 In the **Slice** settings window, locate the **Plane Data** section.

8 In the **Planes** edit field, type 1.

9 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

10 In the **Model Builder** window, right-click **Temperature (nitf)** and choose **Slice**.

11 In the **Slice** settings window, locate the **Plane Data** section.

12 From the **Plane** list, choose **xy-planes**.

13 In the **Planes** edit field, type 8.

14 Locate the **Coloring and Style** section. From the **Color table** list, choose **ThermalLight**.

15 Clear the **Color legend** check box.

16 Click the **Plot** button.