FEMLAB 3

Multiphysics Modeling

Convection Cooling of Circuit Boards solved with FEMLAB 3

© COPYRIGHT 2004 by COMSOL AB. All right reserved.
No part of this documentation may be photocopied or reproduced in any form without prior written consent from COMSOL AB.
FEMLAB is a registered trademark of COMSOL AB. Other product or brand names are trademarks or registered trademarks of their respective holders.

COMSOL

Convection Cooling of Circuit Boards

This discussion models the air cooling of circuit boards populated with multiple integrated circuits (ICs), which act as heat sources. It provides two examples as depicted in Figure 2-1: vertically aligned boards using natural convection, and horizontal boards with forced convection (fan cooling). Convective contributions dominate the cooling. The convection is caused by the induced, or forced, flow of air. To achieve high accuracy, the simulation models heat transport in combination with the fluid flow.

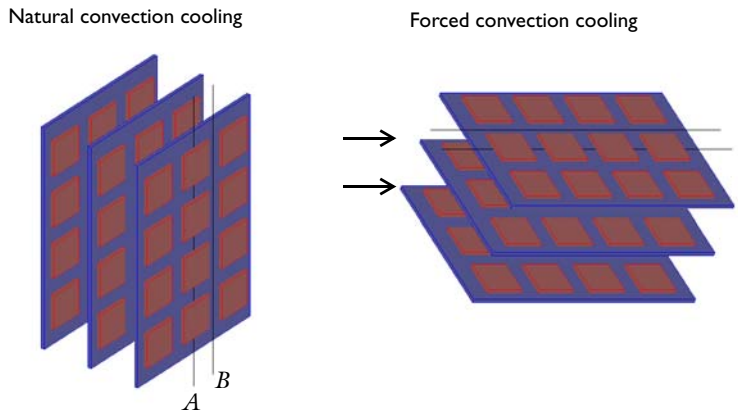

Figure 0-1: Stacked circuit boards with multiple in-row heat sources. Line A represents the center line of the row of ICs, and the area between lines A-B on the board represents the symmetry.

A good 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. For many systems and conditions suffer from a lack of detailed knowledge of h, making accurate calculations of convective heat transfer difficult to achieve.

Instead of simplifying the equations, an alternate 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 achieve accurate determinations of the film

coefficients. These types of models are somewhat more complex but are useful for unusual geometries and complex systems such as circuit-board cooling.

The following examples model the heat transfer of a circuit-board assembly using two application modes: General Heat Transfer and Non-Isothermal Flow. The modeled scenario is based on work published by A. Ortega [1], and this discussion also compares model results with Ortega's experimental results. The first example simulates natural convection cooling of a vertical circuit board, as depicted in Figure 2-1.

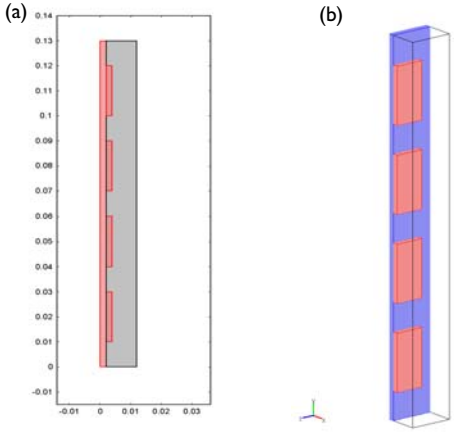

Figure 0-2: The modeled 2D geometry (a), and 3D geometry (b).

In doing the modeling, it is a good idea to first set up a 2D model for the natural convection case. The geometry is the cross section, from the board backside to the next board backside, through the center of a row of ICs (that is, as indicated by line A in Figure 2-1). Next create a 3D model for the same case. Due to symmetry, it is sufficient to model a unit cell, from the backside of a board to the next backside, covering the area between lines A and B in Figure 2-1. Figure 2-2 depicts the two geometries for the natural convection case.

The dimensions of the original geometry are:

- Board: length (in the flow direction) 0.13 m, and the thickness is 0.002 m
- ICs: length and width are both 0.02 m, and thickness is 0.002 m
- The distance of air between the boards is 0.010 m

For the forced convection case, the 3D model is set up by rotating the geometry of Figure 2-2 (b), so that the boards are aligned horizontally.

Model Definition

The model makes use of two stationary application modes to simulate the problem: General Heat Transfer, and Non-Isothermal Flow.

The Non-Isothermal Flow case, modeled with the Navier-Stokes equations, describes the fluid velocity, $\mathbf{u}$, and the pressure, p as:

$$-\nabla \cdot \eta\,(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} + \nabla p = (\rho - \rho_0)\mathbf{g}$$
$$\nabla \cdot (\rho\mathbf{u}) = 0$$

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 $(\rho\text{-}\rho_0)\mathbf{g}$. The model also treats viscosity, η, as being temperature dependent.

The General Heat Transfer application mode is based on a general energy balance

$$\nabla \cdot (-k\nabla T) = Q - (\rho C_P \mathbf{u} \cdot \nabla T)$$

where k represents thermal conductivity; C_p is the specific heat capacity; and Q is the heating power per unit volume, set to 1.25 MW/m^3 (1 W/component) for the 3D cases. For the 2D cases, you should set it to 2/3 of that value to represent the lateral average heating power (that is, taking into account the open slots between the ICs). The material properties appear in Table 2-1.

TABLE 0-1: MATERIAL PROPERTIES

MATERIAL PROPERTY	HEAT SOURCE (SILICON)	CIRCUIT- BOARD (FR4)[2]
ρ [kg/m^3]	2330	1900
C_p [J/kg K]	703	1369
k [W/m K]	163	0.30

The model treats properties for air as being temperature dependent according to the following equations [3]:

$$\rho = (p_0 M_w)/(RT)$$

with p_0 = 101.3 MPa; M_w = 0.0288 kg/mol; and R = 8.314 J/mol K.

$$C_p = 1100\rho$$

$$\log k = (-3.723 + 0.865\log(T))$$

$$\eta = 6.0\times10^{-6} + 4.0\times10^{-8}T$$

We specify the boundary conditions for the flow inlet as boundary-normal flow with a known velocity field. For the natural convection models, set the inlet velocity to zero. For the forced convection cases, set up a parabolic velocity profile (similar to a fully developed laminar profile), u_y, at the inlet as a function of u_{max}, which equals 1 m/s. In terms of an equation it is:

$$u_y = s(1-s)4u_{max}$$

where s represents the normalized inlet width, a predefined parameter in FEMLAB that spans 0 to 1 along each boundary segment. At the outlet the models all use normal flow/zero pressure. In addition, they apply no-slip conditions at the surfaces of the board and the ICs. At the inlet boundary then fix the temperature to 300 degrees K (room temperature), and at the outlet the models use purely convective heat flux. You should also set the lateral boundaries periodic with respect to temperature, making the temperatures equal on both boundaries at every y value. Finally, the models apply continuity of temperature and heat flux at all interior boundaries.

Results and Discussion

NATURAL CONVECTION

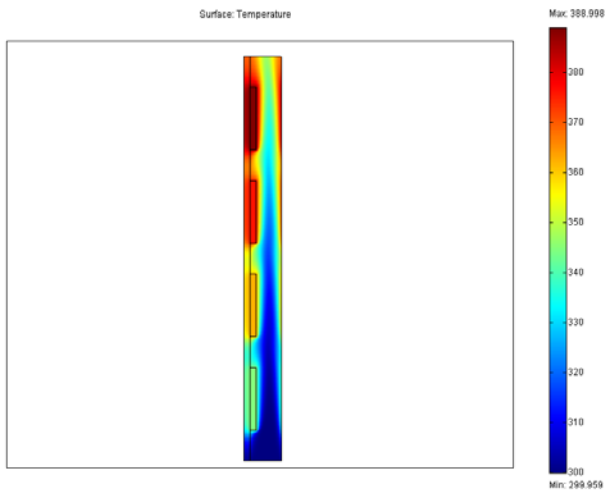

Figure 0-3: Results from the 2D-model in terms of temperature distribution.

The results of the 2D model, depicted in Figure 2-3, show that the temperature of the ICs (heat sources) increases considerably as you apply a heating load of 1 W/ component. Note that the temperature increase of the sources varies from 30 degrees K for the lowest IC up to almost 90 degrees, experienced by the top IC. This is a result of the thermal "footprint" of the heat sources below. 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).

The fluid flow in the 3D problem is a bit more complex to solve because of the increased number of mesh nodes necessary to resolve the flow and heat transport fields. The results, depicted in Figure 2-4, show that the temperature increase at the hottest spot of each component is approximately three degrees higher for the 3D case than for the 2D case.

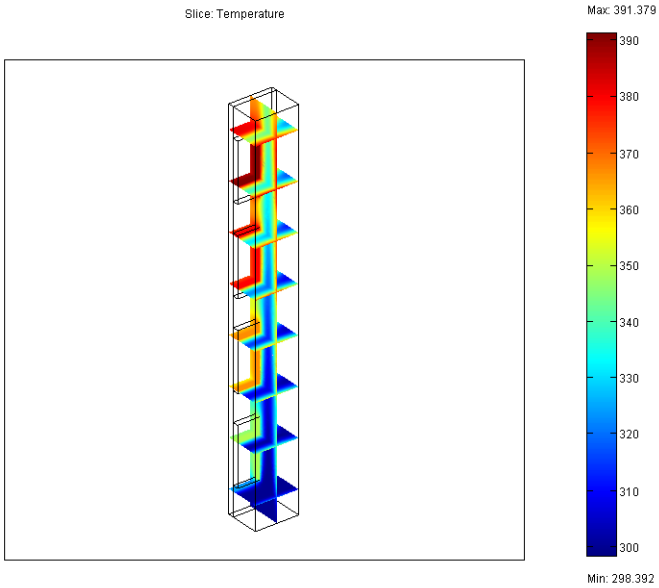

Figure 0-4: Temperature distribution for the 3D modeling results.

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 and 80 degrees K above ambient. This result is probably closer to reality compared to the 2D simulation because it also includes the

horizontal gaps between the ICs. The difference in temperature rise along the board's height is mainly explained by the fluid-flow pattern.

Figure 2-5 plots the fluid velocity for both the 2D and 3D models. The maximum fluid velocity is slightly higher in the 3D case than in the 2D case. More importantly, the flow field behaves differently in the two cases. When comparing Figure 2-5 (a) and (b) note that the velocity fields are rather similar along the center line of the heat sources. However, there is a channeling effect from the horizontal gaps.

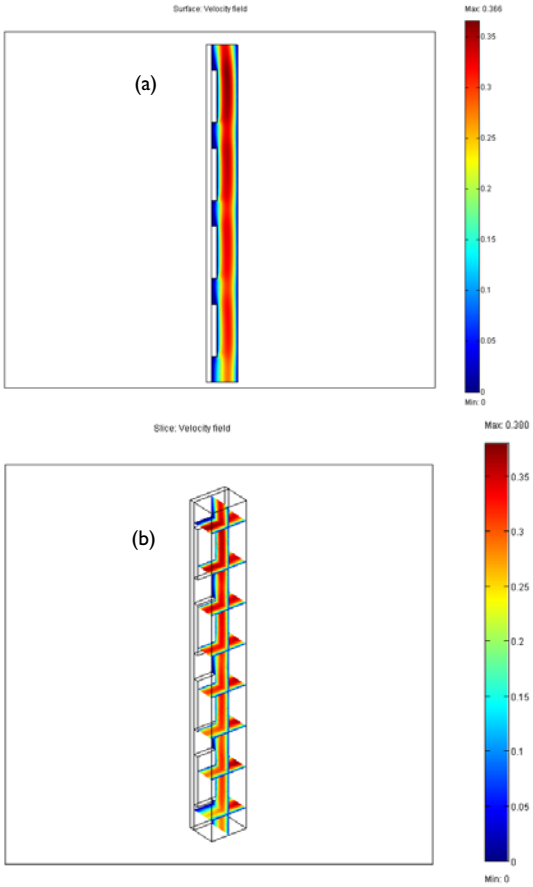

Figure 0-5: Fluid-velocity distribution for the 2D model (a) and the 3D model (b).

FORCED CONVECTION—HORIZONTAL BOARDS

This model includes a forced fluid inlet velocity that represents the situation when a fan cools the ICs. As shown in Figure 2-6, the temperature rise in the ICs is generally somewhat smaller, approximately 50 to 60 degrees, compared to the natural convection cases due to the higher average fluid velocity. In the forced convection case, the difference in temperature rise along the board is more pronounced than in natural convection. This means that the effect of a "thermal footprint" is stronger as the fluid velocity increases.

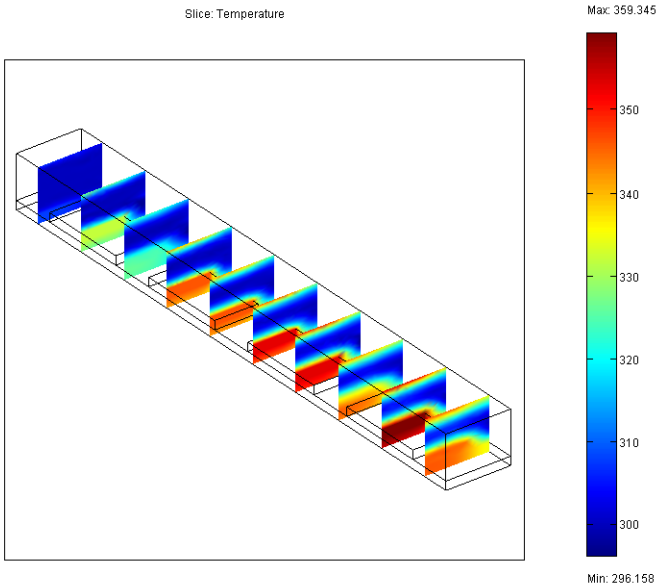

Figure 0-6: Temperature distribution in the case of forced convection cooling of horizontal boards.

Another interesting result, which can be seen in Figure 2-7, is that the channeling effect of the gap causes a reduction in the fluid flow rate above the sources. The cooling of the ICs is therefore reduced somewhat, as compared to an ideal case with an even flow field.

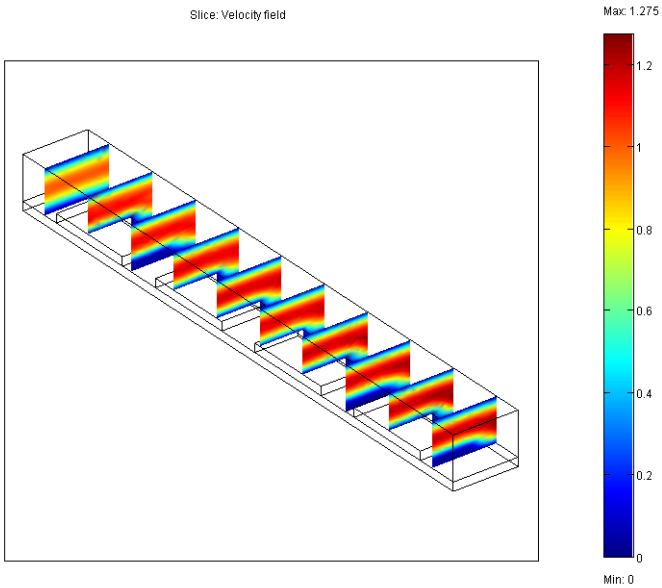

Figure 0-7: Velocity distribution for the case of forced convection.

From the simulation results you can also determine the effective convection heat-transfer film coefficient, h. Calculate it by integrating the heat flux across the fluid boundary of the source objects. Then divide that value with the temperature difference between that of the fluid at the surface and the inlet temperature. Put in terms of an equation, it is:

$$h_i = \left(\frac{1}{\Omega_1}\int q_i \mathrm{d}\Omega_1\right)\left(\frac{1}{\Omega_2}\int T_{\mathrm{s,i}} \mathrm{d}\Omega_2 - \mathrm{T}_{\mathrm{f,0}}\right)^{-1}$$

where Ω_1 is the surface of the source, Ω_2 is the fluid cross section, q_i is the heat flux, while $T_{\mathrm{f,0}}$ and $T_{\mathrm{s,i}}$ represent the inlet fluid temperature and the surface temperature of source i. Thus, the value of h varies between the rows of the sources due to "thermal footprints" from upstream heat sources.

In the case of forced convection, it is common to use the adiabatic film resistance, h_{ad}. It is defined similar to h except it uses T_{cup} instead of $T_{f,0}$. T_{cup} is the cross-section average of the fluid temperature, T_f, upstream of each source, defined as follows:

$$T_{\text{cup}} = \left(\int((\rho\mathbf{n}\cdot u)T_f d\Omega_2)\right)\left(\int(\rho\mathbf{n}\cdot u)d\Omega_2\right)^{-1}$$

Figure 2-8 compares the calculated values of h and h_{ad} for the convection cases with experimentally achieved values using similar geometries.

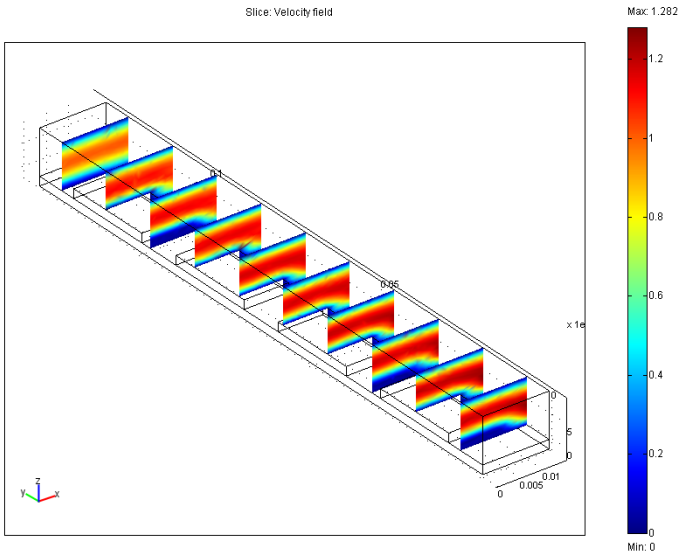

Figure 0-8: Comparison of experimentally measured film resistances, h, (asterisk, solid line) for natural convection[1] with those calculated from the 2D model (squares, dash-dot) and the 3D model (triangles, dashed line).

The deviation from the experimental values for the natural convection may stem from differences in the geometry (which is not fully defined in the publication [1]).

In the forced-convection case you can compare the achieved results with experimental results by calculating the Nusselt number, Nu. It follows from:

$$Nu_L = h_{ad}(L/k)$$

where L in this case is the length of the heat source (20 mm). The calculated Nusselt numbers for the 2D model decrease from 16 to 11 between rows 1 and 4. These values agree well with the experimentally measured ones for similar geometries, being in the range of 15 [1].

A general conclusion you can draw from this example is that modeling can achieve accurate values of convective heat-transfer film coefficients, although the values do differ somewhat between the 2D and 3D models. In addition, the good agreement between experimental values and those from the models indicates the models' high accuracy.

References

[1] A. Ortega, "Air Cooling of Electronics: A Personal Perspective 1981-2001", presentation material from 2002 IEEE SEMITHERM Symposium, March 2002, (`http://info-center.ccit.arizona.edu/~thermlab/publications/ortega-thermie2002.pdf`).

[2] C. Bailey, "Modelling the Effect of Temperature on Product Reliability", Proceedings of the 19th IEEE SEMITHERM Symposium, 2003.

[3] See appendix of J.M. Coulson and J.F. Richardsson, "Chemical Engineering", Vol 1, Pergamon Press, Oxford, U.K., 1990.

Modeling Using the Graphical User Interface—2D Natural Convection

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/circuit_board_nat_2D`

MODEL NAVIGATOR

1 Start FEMLAB.

2 On the **New** page of the **Model Navigator**, set **Space dimension** to **2D.**

3 Click **Multiphysics**. Select the application mode **Heat Transfer Module>General Heat Transfer**, and click **Add**.

4 Select the application mode **Heat Transfer Module>Non-Isothermal Flow**, click **Add**, and click **OK**.

OPTIONS AND SETTINGS

1 Define the following constant expressions in **Constants** under the **Options** menu:

NAME	EXPRESSION
q_source	2/3*1/(2e-3*20e-3^2)
T0	300
rho0_air	1.013e5*28.8e-3/(8.314*T0)

2 Click **OK**.

3 Define the following expressions in **Expressions>Scalar expressions** in the **Options** menu:

NAME	EXPRESSION
k_air	10^(-3.723+0.865*log10(abs(T)))
rho_air	1.013e5*28.8e-3/(8.314*T)
Cp_air	1.1e3*rho_air
eta_air	6e-6+4e-8*T

4 Click **OK**.

GEOMETRY MODELING

1 Create rectangles according to the following table by using **Specify Object>Rectangle** in the **Draw** menu.

OBJECT	WIDTH	HEIGHT	BASE	X	Y
R1	0.002	0.13	Corner	0	0
R2	0.01	0.13	Corner	0.002	0
R3	0.002	0.02	Corner	0.002	0.01

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

3 With R3 selected click the **Array** toolbar button, type 0.03 in the **y** edit field in the **Displacement area** and type 4 in the **y** edit field in the **Array size** area. Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 In the **Multiphysics** menu, select the **Non-Isothermal Flow** application mode.

2 Open **Subdomain Settings** in the **Physics** menu.

3 Select subdomains 1 and 3–6. Click the **Active in this domain** check box to deactivate the **Non-Isothermal Flow** application mode in those subdomains.

4 Select subdomain 2 and enter the following expressions in the edit fields:

PARAMETER	EXPRESSION
ρ	rho_air
η	eta_air
F_y	9.81*(rho0_air-rho_air)

5 Click **OK** to close the **Subdomain Settings** dialog box.

6 In the **Multiphysics** menu, select the **General Heat Transfer** application mode.

7 Open the **Subdomain Settings** dialog box and select all subdomains. On the **Init** page type `T0` in the **Temperature** edit field.

8 Apply the following settings on the **Conduction** page.

SUBDOMAIN	1	2
k (isotropic)	`0.3`	`k_air`
ρ	`1900`	`rho_air`
C_p	`1369`	`Cp_air`

9 With subdomain 2 selected, click the **Convection** tab, click the **Enable convective heat transfer** check box, and type `u` and `v` in the edit fields for **x-velocity** and **y-velocity**.

10 On the **Conduction** page, select subdomains 3-6. Click the **Load** button, choose **Silicon**, and click **OK**.

11 In the **Q** edit field, type `q_source`.

12 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 Open the **Boundary Settings** dialog box from the **Physics** menu.

2 Set the boundary conditions according to the following table.

BOUNDARY	5	22
Boundary condition	Temperature	Convective flux
T_0	`T0`	
Radiation type	`None`	

3 Click **OK**.

4 In the **Multiphysics** menu, select **Non-Isothermal Flow**. Open the **Boundary Settings** dialog box.

5 Select all boundaries and apply the **No slip** boundary condition.

6 Select boundaries 5 and 22, apply **Normal flow>Pressure** conditions with pressure 0.

7 Click **OK**.

8 Open the **Periodic Boundary Conditions** dialog box from the **Physics>Periodic Conditions** menu.

9 On the **Source** page, select boundary 1 and type `T` in the **Expression** edit field, and press **Enter**.

10 Click the **Destination** tab, select boundary 27 by selecting the corresponding check box, and type `T` in the **Expression** edit field.

11 Click the **Source Vertices** tab, select boundaries 1 and 2 in the **Vortex selection** list and click the **>>** button.

12 Click the **Destination Vertices** tab and select and add vertices 21 and 22. Click **OK**.

MESH GENERATION

1 Open the **Mesh Parameters** dialog box from the **Mesh** menu. On the **Global** page select **Normal** from the **Predefined mesh sizes** list.

2 On the **Subdomain** page, select subdomain 2 and type `1.5e-3` in the **Maximum element size** edit field. Click **OK**.

3 Click **Initialize Mesh** in the **Mesh** menu.

COMPUTING THE SOLUTION

1 Click the **Solve** button to calculate the solution.

POSTPROCESSING AND VISUALIZATION

1 In order to create Figure 2-3 open the **Plot Parameters** dialog box from the **Postprocessing** menu.

2 On the **Surface** page select **Temperature** from the list of **Predefined quantities** in the **Surface data** area. Click **OK**.

3 To achieve Figure 2-5 (a) change the **Surface data** to **Velocity field** similar to above and click **OK**.

Modeling Using the Graphical User Interface—3D Natural Convection

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/circuit_board_nat_3D`

1 Repeat the steps from the 2D model regarding the *Model Navigator* and *Options And Settings*, however, select **3D** in the **Space dimension** list.

GEOMETRY MODELING

1 Create blocks according to the following table by using **Block** in the **Draw** menu.

OBJECT	LENGTH X	LENGTH Y	LENGTH Z	BASE X	BASE Y	BASE Z
BLK1	0.015	0.002	0.13	0	0	0
BLK2	0.01	0.002	0.02	0	-0.002	0.01
BLK3	0.015	0.01	0.13	0	-0.01	0

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

3 With **BLK2** selected click the **Array** toolbar button, type `0.03` in the **z** edit field in the **Displacement** area and type `4` in the **z** edit field in the **Array size** area. Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 Open **Subdomain Settings** in the **Physics** menu.

2 Select subdomains 2 to 6. Click to clear the **Active in this subdomain** check box to deactivate the **Non-Isothermal Flow** application mode in those subdomains.

3 Select subdomain 1 and enter the following expressions in the edit fields:

PARAMETER	EXPRESSION
ρ	`rho_air`
η	`eta_air`
F_z	`9.81*(rho0_air-rho_air)`

4 Click the **Artificial Diffusion** button, select **Streamline diffusion**, and click **OK**.

5 Click **OK** to close the **Subdomain Settings** dialog box.

6 In the **Multiphysics** menu, select the **General Heat Transfer** application mode.

7 Open the **Subdomain Settings** dialog box and select all subdomains. Click the **Init** tab and type `T0` in the **Temperature** edit field.

8 Apply the following settings on the **Conduction** page

SUBDOMAIN	1	6
k (isotropic)	`k_air`	`0.3`
ρ	`rho_air`	`1900`
C_p	`Cp_air`	`1369`

9 With subdomain 1 selected, click the **Convection** tab, click **Enable convective heat transfer** and type `u`, `v` and `w` in the edit fields for **x-**, **y-** and **z-velocity**.

10 Click the **Artificial Diffusion** button, select **Streamline diffusion**, and click **OK**.

11 On the **Conduction** page, select subdomains 2-5. Click the **Load** button, choose **Silicon** and click **OK**.

12 In the **Q** edit field, type `q_source`.

13 Click **OK** to close the **Subdomain Settings** dialog box.

Boundary Conditions

1 Open the **Boundary Settings** dialog box from the **Physics** menu.

2 Set the boundary conditions according to the following table.

BOUNDARY	3	4
Boundary condition	Temperature	Convective flux
T_0	`T0`	
Radiation type	`None`	

3 Click **OK**.

4 In the **Multiphysics** menu select **Non-Isothermal Flow**. Open the **Boundary Settings** dialog box.

5 Select all boundaries and apply the **No slip** boundary condition.

6 Select boundaries 3 and 4, apply the **Normal flow/Pressure** boundary condition with pressure 0.

7 Select boundaries 1 and 34, apply the **Slip/Symmetry** boundary condition.

8 Click **OK**.

9 Open the **Periodic Boundary Conditions** dialog from the **Physics>Periodic Conditions** menu.

10 On the **Source** page, select boundary 2 and type `T` in the **Expression** edit field, and press **Enter**.

11 Click the **Destination** tab, select boundary 29 by selecting the corresponding check box, and type `T` in the **Expression** field.

12 In the **Source Vertices** page select and add vertices 1, 2, 39, and 40, in the mentioned order, by selecting them and clicking the "**>>**" button.

13 In the **Destination Vertices** page select and add vertices 21, 22, 43, and 44, also in the mentioned order. Click **OK**.

MESH GENERATION

1. Open the **Mesh Parameters** dialog box from the **Mesh** menu. On the **Global** page select **Coarser** from the **Predefined mesh sizes** list.
2. On the **Boundary** page, select boundary 2, type `3e-3` in the **Maximum element size** edit field and type `2` in the **Element growth rate** edit field.
3. On the **Advanced** page, type `0.5` in the edit field for **x-direction scale factor.** Click **OK**.
4. Click **Initialize Mesh** in the **Mesh** menu.

COMPUTING THE SOLUTION

1. Open the **Solve Parameters** dialog box from the **Solve** menu.
2. Select the **Stationary nonlinear** solver and select **Direct (UMFPACK)** from the **Linear system solver** list.
3. Clear the **Symmetric matrices** check box.
4. Click **OK**.
5. Click **Solve** to compute the solution. The problem is rather large, because of the strong coupling between temperature and velocity and because of the dense mesh. To solve the problem approximately 500 MB of free memory is required on the computer. The solution is achieved in approximately 15 minutes (using a 1.5GHz Intel M Processor).

POSTPROCESSING AND VISUALIZATION

1. In order to create Figure 2-4 open the **Plot Parameters** dialog from the **Postprocessing** menu.
2. In the **General** page clear **Slice** and select **Boundary** plot type.
3. In the **Boundary** page select **Temperature** from the list of **Predefined quantities** in the **Surface data** field. Click **Apply**.
4. To produce Figure 2-5 (b), enable **Slice** and disable **Boundary** plot types in the **General page.**
5. Select **Velocity field** as data in the **Slice** page.
6. Type 1, 0 and 8 in the edit fields **x**, **y**, and **z levels.** Click **OK**.

Modeling Using the Graphical User Interface—3D Forced Convection

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/circuit_board_forced_3D`

This model is achieved by modifying the previous model (`circuit_board_nat_3D`). Begin with loading, or building that model.

GEOMETRY MODELING

1 Go to draw mode by clicking the **Draw toolbar button.**

2 Select all objects and open the **Rotate** dialog box from **Draw>Modify**.

3 Enter `-90` in the edit field for **Rotation** angle.

4 Enter `1`, `0`, and `0` in the **x, y** and **z** edit fields for **Rotation axis direction vector**. Click **OK**.

PHYSICS SETTINGS

Boundary Settings

1 In the **Multiphysics** menu, select the **General Heat Transfer** application mode.

2 Open **Boundary Settings** in the **Physics** menu.

3 Set the boundary conditions according to the following table.

BOUNDARY	5	29
Boundary condition	Convective flux	Temperature
T_0		`T0`
Radiation type		`None`

4 Click **OK**.

5 In the **Multiphysics** menu select **Non-Isothermal Flow**. Open the **Boundary Settings** dialog box.

6 Select boundary 29 and apply the **Inflow/Outflow velocity** condition.

7 Enter `-4*(1e4)*(z)*(0.01-(z))*(z>0)` in the edit field for **y-velocity**. Click **OK**.

COMPUTING THE SOLUTION

1 Open the **Solver Manager** dialog box from the **Solve** menu.

2 On the **Init** page, select **Initial value expression** in the **Initial value** area.

3 Go to the **Solve For** page, select the **Non-Isothermal Flow** application mode.

4 Click **Solve**. The solution process takes approximately 7 minutes and requires 500 MB memory.

5 When the solution has been achieved, click the **Init** page of the **Solver Manager.** Select **Current Solution** in the **Initial value** area.

6 Go to the **Solve For** page, select both application modes.

7 Click **Solve** to calculate the final result. This process also takes approximately 7 minutes.

POSTPROCESSING AND VISUALIZATION

1 In order to create Figure 2-6 open the **Plot Parameters** dialog from the **Postprocessing** menu.

2 In the **General** page, select the **Slice** plot type and clear the **Boundary** plot type.

3 In the **Slice** page, select **Temperature** from the list of **Predefined quantities** in the **Surface data** field.

4 Type 1, 10, and 0 in the edit fields **x**, **y**, and **z levels.** Click **Apply**.

5 To produce Figure 2-7, select **Velocity field** as data in the **Slice** page.

6 Type 0 in the **x levels** edit field and click **OK**.

Create the plot shown in the **Model Navigator** and when the model is opened.

7 Open the **Plot Parameters** from the **Postprocessing** menu.

8 Deactivate **Slice** and **Arrow** and activate **Boundary** and **Streamline** on the **General** page.

9 Click on the **Streamline** page.

10 Select **Specify start point coordinates** in the **Streamline start points** area.

11 Enter `linspace(1e-3,13e-3,13)` in the **x** edit field.

12 Enter `linspace(0.13,0.13,13)` in the **y** edit field.

13 Enter `linspace(1.3e-3,1.3e-3,13)` in the **z** edit field.

14 Select **Use expression to color streamlines** in the **Streamline color** area.

15 Click on the **Color Expression** button to open the **Streamline Color Expression** dialog box.

16 Enter `T-296` in the **Expression** field, click **OK** to close the dialog box.

17 Click on the **Tube Radius** button to open the **Tube Radius Parameters** dialog box.

18 Click on the **Radius data** check box and select **Temperature gradient** from the **Predefined quantities** list.

19 Clear the **Radius scale factor Auto** checkbox and enter `0.5` as the **Scale factor**, click **OK** to close the dialog box.

20 Click the **Advanced** button to open the **Advanced Streamline Parameters** dialog box.

21 Enter `1000` in the **Maximum number of integration steps** edit field, click **OK** to close the dialog box.

22 Select **Options>Suppress>Suppress Boundaries** to open the **Suppress Boundaries** dialog box.

23 Select boundary 4, 5, 6, 7, 9, 29, and 35, click **OK** to close the dialog box.

24 Click the **Postprocessing Mode** toolbar button.

25 Click the **Scene Light** toolbar button.

26 Double click the **Axis** field in the status bar at the bottom of the FEMLAB window to disable the axis.