COMSOL MULTIPHYSICS

HEAT TRANSFER MODULE

MODEL LIBRARY

VERSION 3.5

COMSOL

How to contact COMSOL:

Benelux
COMSOL BV
Röntgenlaan 19
2719 DX Zoetermeer
The Netherlands
Phone: +31 (0) 79 363 4230
Fax: +31 (0) 79 361 4212
info@comsol.nl
www.comsol.nl

Denmark
COMSOL A/S
Diplomvej 376
2800 Kgs. Lyngby
Phone: +45 88 70 82 00
Fax: +45 88 70 80 90
info@comsol.dk
www.comsol.dk

Finland
COMSOL OY
Arabianranta 6
FIN-00560 Helsinki
Phone: +358 9 2510 400
Fax: +358 9 2510 4010
info@comsol.fi
www.comsol.fi

France
COMSOL France
WTC, 5 pl. Robert Schuman
F-38000 Grenoble
Phone: +33 (0)4 76 46 49 01
Fax: +33 (0)4 76 46 07 42
info@comsol.fr
www.comsol.fr

Germany
COMSOL Multiphysics GmbH
Berliner Str. 4
D-37073 Göttingen
Phone: +49-551-99721-0
Fax: +49-551-99721-29
info@comsol.de
www.comsol.de

Italy
COMSOL S.r.l.
Via Vittorio Emanuele II, 22
25122 Brescia
Phone: +39-030-3793800
Fax: +39-030-3793899
info.it@comsol.com
www.it.comsol.com

Norway
COMSOL AS
Søndre gate 7
NO-7485 Trondheim
Phone: +47 73 84 24 00
Fax: +47 73 84 24 01
info@comsol.no
www.comsol.no

Sweden
COMSOL AB
Tegnérgatan 23
SE-111 40 Stockholm
Phone: +46 8 412 95 00
Fax: +46 8 412 95 10
info@comsol.se
www.comsol.se

Switzerland
FEMLAB GmbH
Technoparkstrasse 1
CH-8005 Zürich
Phone: +41 (0)44 445 2140
Fax: +41 (0)44 445 2141
info@femlab.ch
www.femlab.ch

United Kingdom
COMSOL Ltd.
UH Innovation Centre
College Lane
Hatfield
Hertfordshire AL10 9AB
Phone:+44-(0)-1707 636020
Fax: +44-(0)-1707 284746
info.uk@comsol.com
www.uk.comsol.com

United States
COMSOL, Inc.
1 New England Executive Park
Suite 350
Burlington, MA 01803
Phone: +1-781-273-3322
Fax: +1-781-273-6603

COMSOL, Inc.
10850 Wilshire Boulevard
Suite 800
Los Angeles, CA 90024
Phone: +1-310-441-4800
Fax: +1-310-441-0868

COMSOL, Inc.
744 Cowper Street
Palo Alto, CA 94301
Phone: +1-650-324-9935
Fax: +1-650-324-9936

info@comsol.com
www.comsol.com

For a complete list of international representatives, visit www.comsol.com/contact

Company home page
www.comsol.com

COMSOL user forums
www.comsol.com/support/forums

Heat Transfer Module Model Library

Version: September 2008 COMSOL 3.5

Part number: CM020802

C O N T E N T S

Chapter 1: Introduction

Chapter 2: Electronics and Power-System Models

Chapter 3: Processing and Manufacturing Models

Chapter 4: Medical Technology Models

1

Introduction

The *Heat Transfer Module Model Library* consists of a set of models that simulate problems in various areas of heat transfer and other engineering disciplines where heat transfer plays an important role. Their purpose is to assist you in learning, by example, how to model sophisticated heat transfer processes. Through them you can tap the expertise of the top researchers in the field, examining how they approach some of the most difficult modeling problems you might encounter. You can thus get a feel for the power that COMSOL Multiphysics® offers as a modeling tool. In addition to serving as a reference, the models can also give you a big head start if you are developing a model of a similar nature.

This book divides these models into three chapters:

- Electronics and power systems
- Process and manufacturing
- Medical technology

The models illustrate the application modes specific to the Heat Transfer Module, application modes unavailable in the base COMSOL Multiphysics package. These application modes come with their own graphical user interfaces that make it quick and easy to access their power. You can even modify them for custom requirements. COMSOL Multiphysics itself is very powerful and, with sufficient expertise in a given field, you certainly could develop these modes by yourself—but why spend the hundreds or thousands of hours that would be necessary when our team of experts has already done the work for you?

Note that the model descriptions in this book do not contain every detail on how to carry out every step in the modeling process. Before tackling these in-depth models, we urge you to first read the other book in the Heat Transfer Module documentation set. Titled the *Heat Transfer Module User's Guide*, it introduces you to the basic functionality in the module, covers basic modeling techniques, and includes reference material of interest to those working with problems involving heat transfer. The models it presents are far simpler than those in this Model Library and might be more appropriate for a first introduction to COMSOL Multiphysics.

In addition, to gain further information on how to work with the graphical user interface you can turn to the *COMSOL Multiphysics User's Guide* or the *COMSOL Multiphysics Quick Start and Quick Reference* manual. An explanation on how to perform modeling with a programming language is available in the *COMSOL Multiphysics Scripting Guide*.

This *Heat Transfer Module Model Library* provides details about a large number of ready-to-run models that illustrate real-world uses of the software. Each entry comes with theoretical background as well as instructions that illustrate how to set it up. They come from our staff engineers, who have years of experience in heat transfer modeling. The terminology in the book should be familiar to you.

Finally, each example in the *Heat Transfer Module Model Library* as well as in the *Heat Transfer Module User's Guide* comes with the software as a loadable COMSOL Multiphysics Model MPH-file (with the extension `.mph`). To find these files, start the **Model Navigator**, click the **Model Library** tab and then look under the chapter headings listed earlier. These models are great to investigate if you are sufficiently familiar with COMSOL Multiphysics and its GUI but would like to learn more about how to set up a certain model. You can even use these entries as a starting point for your own models that are similar in nature.

Model Library Guide

The following table summarizes key information about the entries in this model library. The "Application Modes" column indicates which modes we chose to solve the model, and the subsequent column indicates the number of spatial dimensions in the model.

The solution time given is the elapsed time measured on a machine running Windows Vista with a 2.6 GHz AMD Athlon X2 Dual Core 500 CPU and 2 GB of RAM. For models with a sequential solution strategy, the Solution Time column shows the total solution time for all solution steps.

The next columns indicate if the model is stationary or time-dependent, which heat transfer mechanisms are involved, and if the model includes multiphysics.

MODEL	PAGE	APPLICATION MODES	SPATIAL DIMENSIONS	SOLUTION TIME	STATIC	TIME DEPENDENT	CONDUCTION	CONVECTION	RADIATION	OUT-OF-PLANE	HIGHLY CONDUCTIVE LAYER	MULTIPHYSICS
ELECTRONICS AND POWER SYSTEMS												
Convection Cooling of Circuit Boards, Forced 3D Model	10	General Heat Transfer, Weakly Compressible Navier-Stokes	3D	6 min	√		√	√				√
Convection Cooling of Circuit Boards, Natural 2D Model	10	General Heat Transfer, Weakly Compressible Navier-Stokes	2D	20 s	√		√	√				√
Convection Cooling of Circuit Boards, Natural 3D Model	10	General Heat Transfer, Weakly Compressible Navier-Stokes	3D	4 min	√		√	√				√
Convection Cooling of Circuit Boards, Simplified 1D Model	31	General Heat Transfer	1D	1 s	√		√	√				
Convection Cooling of Circuit Boards, Simplified 3D Model	31	General Heat Transfer	3D	6 s		√	√					
Forced Turbulent Convection Cooling	43	General Heat Transfer, k-ε Turbulence Model	2D	16 min	√		√	√				√
Forced Turbulent Convection Cooling, Simplified Model	43	General Heat Transfer	2D	1 s	√		√					
Surface-Mount Resistor	63	General Heat Transfer, Solid Stress-Strain	3D	2 min	√		√					√
Free Convection in a Light Bulb‡	78	General Heat Transfer Weakly Compressible Navier-Stokes,	2D-axi	6 min		√	√	√	√			√

MODEL	PAGE	APPLICATION MODES	SPATIAL DIMENSIONS	SOLUTION TIME	STATIC	TIME DEPENDENT	CONDUCTION	CONVECTION	RADIATION	OUT-OF-PLANE	HIGHLY CONDUCTIVE LAYER	MULTIPHYSICS
Heating Circuit	91	General Heat Transfer, Thin Conductive Shell, Solid Stress-Strain, Shell	3D	4 min	√		√				√	√
Microchannel Heat Sink	107	General Heat Transfer, Weakly Compressible Navier-Stokes	3D	2 min	√		√	√				√
Microchannel Heat Sink, Resistance	107	General Heat Transfer, Weakly Compressible Navier-Stokes	3D	2 min	√		√	√				√
Heat Transfer in a Surface-Mount Package for a Silicon Chip	123	General Heat Transfer	3D	1 s	√		√				√	
Rapid Thermal Annealing	135	General Heat Transfer	3D	9 s		√	√		√			
Thermo-Photo-Voltaic Cell	144	General Heat Transfer	2D	2 min	√		√		√			
Convective Cooling of a Potcore Inductor	160	General Heat Transfer, Weakly Compressible Navier-Stokes	2D axi	2 min	√		√	√	√			√
Temperature Distribution in a Disc-Type Transformer	169	General Heat Transfer, Weakly Compressible Navier-Stokes	2D axi	2 min	√		√	√				√
PROCESS AND MANUFACTURING												
Heat Generation in a Disc Brake	184	General Heat Transfer; Weak Form, Point	3D	2 min		√	√	√	√			
Convection Cooking of Chicken Patties	197	General Heat Transfer, Diffusion	2D axi	2 s		√	√					√
Continuous Casting	234	General Heat Transfer, Weakly Compressible Navier-Stokes	2D axi	2 min	√		√	√				√
Cooling Flange	209	General Heat Transfer	3D	41 s	√		√					
Friction Stir Welding	223	General Heat Transfer	3D	37 s	√		√	√	√			

MODEL	PAGE	APPLICATION MODES	SPATIAL DIMENSIONS	SOLUTION TIME	STATIC	TIME DEPENDENT	CONDUCTION	CONVECTION	RADIATION	OUT-OF-PLANE	HIGHLY CONDUCTIVE LAYER	MULTIPHYSICS
Turbulent Flow Through a Shell-and-Tube Heat Exchanger	248	General Heat Transfer, k-ω Turbulence Model	2D	2 min	√			√				√
Fluid-Structure Interaction in Aluminum Extrusion†‡	262	General Heat Transfer; Non-Newtonian Flow; Solid, Stress-Strain	3D	30 min	√		√	√			√	√
MEDICAL TECHNOLOGY												
Microwave Cancer Therapy	278	Bioheat Equation, TM Waves	3D	17 s		√	√					√
Tumor Removal	291	Bioheat Equation, Conductive Media DC	3D	38 s		√	√					√
TUTORIAL MODELS												
Radiation in a Cavity	131*	General Heat Transfer	2D	4 s	√		√		√			
Copper Layer on Silica Glass	142*	General Heat Transfer	2D	1 s		√	√				√	
Copper Layer on Silica Glass, Meshed	142*	General Heat Transfer	2D	3 s		√	√					
Conduction in a Cylinder	87*	General Heat Transfer	2D	1 s	√		√					
Displacement Ventilations	248*	General Heat Transfer, k-ε Turbulence Model	3D	100 min	√		√	√				√
Heat Exchanger	91*	General Heat Transfer	3D	7 s	√		√	√				
Heat Exchanger, Nonisothermal	91*	General Heat Transfer, Weakly Compressible Navier-Stokes	3D	2 min	√		√	√				√
Heated Plate	231*	General Heat Transfer, Weakly Compressible Navier-Stokes	2D	3 min	√		√	√				√
Shell Conduction	163*	Thin Conductive Shell	3D	1 s	√		√					
Natural Convection Cooling in a Thermos, Heat Transfer Coefficient Model	109*	General Heat Transfer	2D axi	1 s	√		√					

MODEL	PAGE	APPLICATION MODES	SPATIAL DIMENSIONS	SOLUTION TIME	STATIC	TIME DEPENDENT	CONDUCTION	CONVECTION	RADIATION	OUT-OF-PLANE	HIGHLY CONDUCTIVE LAYER	MULTIPHYSICS
Natural Convection Cooling in a Thermos, Laminar Flow Model	109*	General Heat Transfer, Weakly Compressible Navier-Stokes	2D axi	34 s	√		√	√				√
Thin plate, 2D Model	146*	General Heat Transfer	2D	1 s	√		√		√	√		
Thin plate, 3D Model	146*	General Heat Transfer	3D	3 s	√		√		√	√		

* This page number refers to the *Heat Transfer Module User's Guide*.
† This model requires the Chemical Engineering Module and the Structural Mechanics Module.
‡ This model uses the Material Library.

We welcome any questions, comments or suggestions you might have concerning these models. Contact us at `info@comsol.com`.

Typographical Conventions

All COMSOL manuals use a set of consistent typographical conventions that should make it easy for you to follow the discussion, realize what you can expect to see on the screen, and know which data you must enter into various data-entry fields. In particular, you should be aware of these conventions:

- A **boldface** font of the shown size and style indicates that the given word(s) appear exactly that way on the COMSOL graphical user interface (for toolbar buttons in the corresponding tooltip). For instance, we often refer to the **Model Navigator**, which is the window that appears when you start a new modeling session in COMSOL; the corresponding window on the screen has the title **Model Navigator**. As another example, the instructions might say to click the **Multiphysics** button, and the boldface font indicates that you can expect to see a button with that exact label on the COMSOL user interface.
- The names of other items on the graphical user interface that do not have direct labels contain a leading uppercase letter. For instance, we often refer to the Draw toolbar; this vertical bar containing many icons appears on the left side of the user

interface during geometry modeling. However, nowhere on the screen will you see the term “Draw” referring to this toolbar (if it were on the screen, we would print it in this manual as the **Draw** menu).

- The symbol **>** indicates a menu item or an item in a folder in the **Model Navigator**. For example, **Physics>Equation System>Subdomain Settings** is equivalent to: On the **Physics** menu, point to **Equation System** and then click **Subdomain Settings**. **COMSOL Multiphysics>Heat Transfer>Conduction** means: Open the **COMSOL Multiphysics** folder, open the **Heat Transfer** folder, and select **Conduction**.
- A `Code` (monospace) font indicates keyboard entries in the user interface. You might see an instruction such as “Type `1.25` in the **Current density** edit field.” The monospace font also indicates COMSOL Script codes.
- An *italic* font indicates the introduction of important terminology. Expect to find an explanation in the same paragraph or in the Glossary. The names of books in the COMSOL documentation set also appear using an italic font.

2

Electronics and Power-System Models

This chapter contains models of heat transfer in application such as electronic cooling and power systems.

Convection Cooling of Circuit Boards

Introduction

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 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.

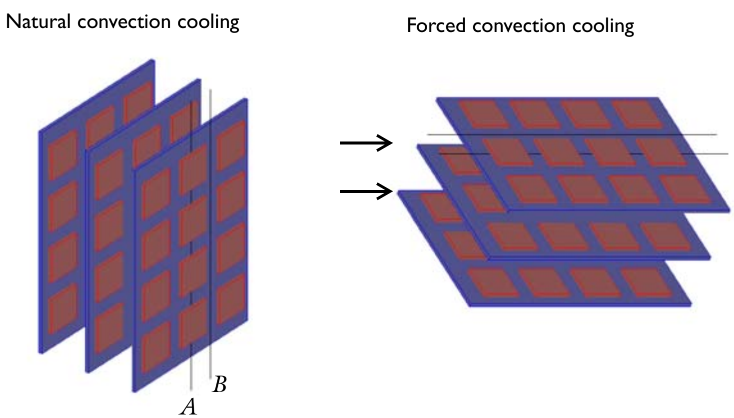


Figure 2-1: Stacked circuit boards with multiple in-line heat sources. Line A represents the center line of the row of ICs, and the area between lines A-B on the board represents the symmetry.

A 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 be well determined. Many systems and conditions suffer from a lack of detailed knowledge of h, making accurate calculations of convective heat transfer difficult.

Instead of simplifying the equations, 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 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 Weakly Compressible Navier-Stokes. The modeled scenario is based on work published by A. Ortega (Ref. 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.

It is a good idea to first set up a 2D model for the case of natural convection. The geometry is the 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 2-1). Next create a 3D model for the same case. 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 2-1. Figure 2-2 depicts the two geometries for the case of natural convection.

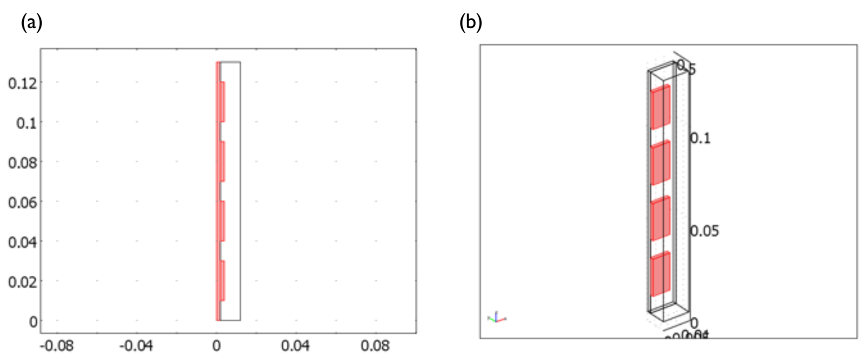


Figure 2-2: The modeled geometries in 2D (a) and 3D (b).

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, set up the 3D model 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 Weakly Compressible Navier-Stokes.

The Weakly Compressible Navier-Stokes, describes the fluid velocity, $\mathbf{u}$, and the pressure, p as

$$\rho\mathbf{u}\cdot\nabla\mathbf{u} = \nabla\cdot[-p\mathbf{I}+\eta(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)-(2\eta/3-\kappa)(\nabla\cdot\mathbf{u})\mathbf{I}]+(\rho-\rho_0)\mathbf{g}$$
$$\nabla\cdot(\rho\mathbf{u}) = 0$$

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

The General Heat Transfer application mode is based on the 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 $\mathrm{MW/m^3}$ (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 2-1: MATERIAL PROPERTIES

MATERIAL PROPERTY	HEAT SOURCE (SILICON)	CIRCUIT BOARD (FR4; REF. 2)
ρ ($\mathrm{kg/m^3}$)	2330	1900
C_p (J/(kg·K))	703	1369
k (W/(m·K))	163	0.30

The model treats properties for air as temperature dependent according to the following equations (Ref. 3):

$$\rho = (p+p_0)M_\mathrm{w}/(RT)$$

with $p_0 = 101.3$ kPa, $M_\mathrm{w} = 0.0288$ kg/mol, and $R = 8.314$ J/(mol·K). Further,

$$C_p = 1100\ \mathrm{J/(kg\cdot K)}$$

$$k = 10^{-3.723\ +\ 0.865\log_{10}(T)}\ \mathrm{W/(m\cdot K)}$$

$$\eta = 6.0 \cdot 10^{-6} + 4.0 \cdot 10^{-8} T \quad \text{Pa·s}$$

where T must be expressed in kelvin.

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 an inlet-velocity profile, u_y, that is uniform in the (horizontal) x direction and parabolic (similar to a fully developed laminar profile) in the (vertical) z direction. In terms of an equation this reads

$$u_y = 4\tilde{z}(1-\tilde{z})(-u_{\max})$$

where $\tilde{z} = z/0.010\ \text{m}$ parameterizes the height above the board and $u_{\max}$, the maximal inlet speed, equals $1\ \text{m/s}$. At the outlet all the models use the normal flow, zero pressure boundary condition. 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 K (room temperature). 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.

NATURAL CONVECTION

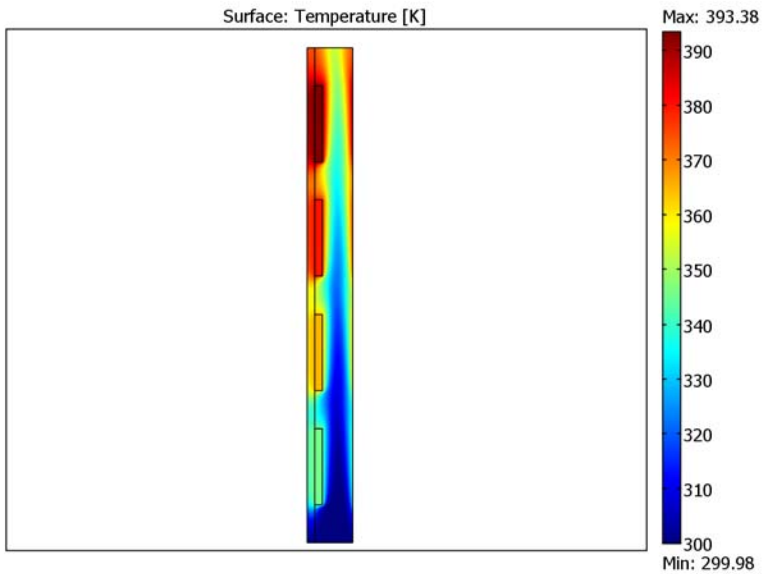


Figure 2-3: Temperature distribution for the 2D model.

The results of the 2D model (Figure 2-3) show that the temperature of the ICs (the heat sources) increases considerably under a heating load of 1 W/component. Note that the temperature increase of the sources varies from 30 K for the lowest IC up to almost 90 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).

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 (Figure 2-4) show that the temperature increase at the hottest spot of each component is approximately two degrees higher for the 3D case than for the 2D case.

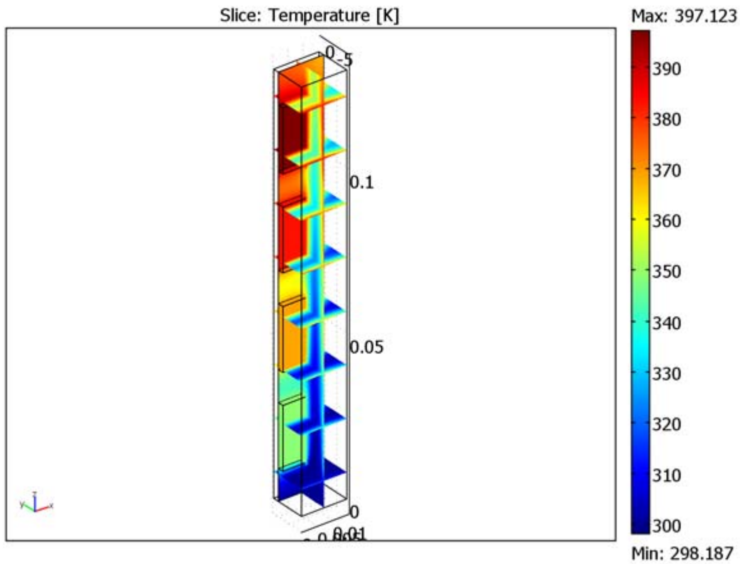


Figure 2-4: Temperature distribution for the 3D model.

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 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 explained primarily 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.

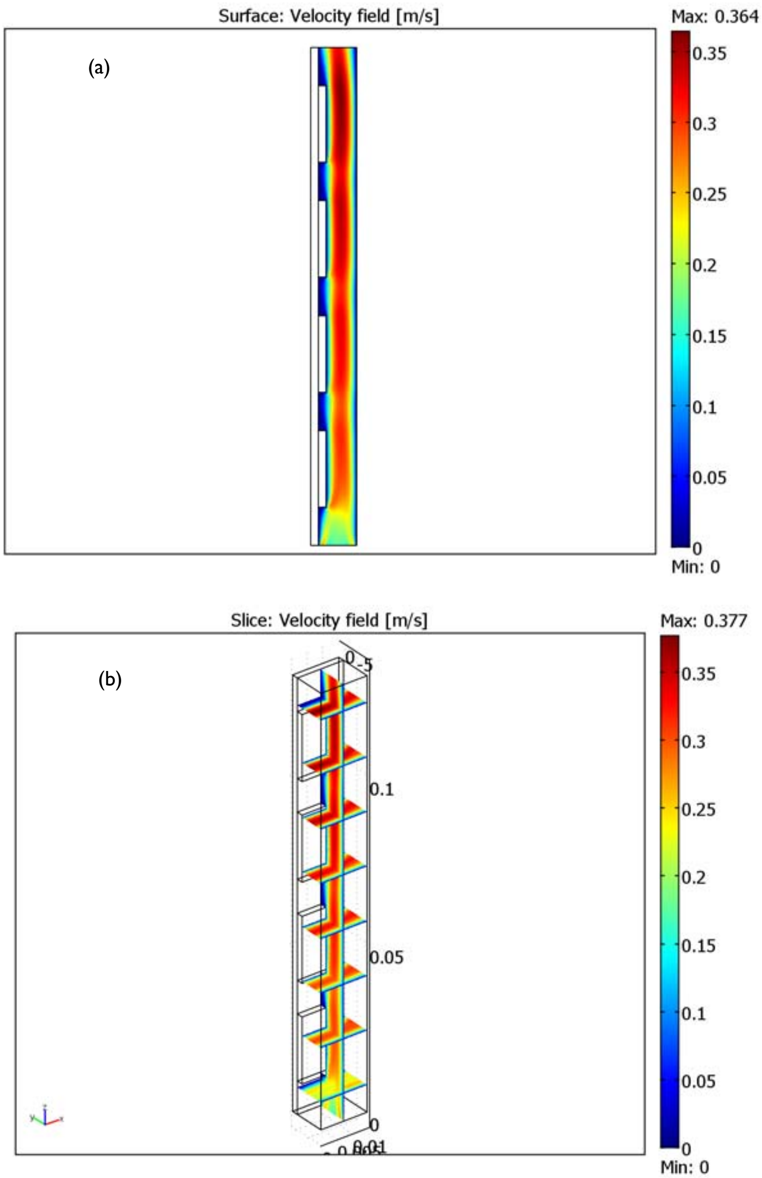


Figure 2-5: Fluid-velocity distribution for the 2D model (a) and the 3D model (b).

FORCED CONVECTION—HORIZONTAL BOARDS

This model includes a forced fluid inlet velocity that represents the situation when a fan cools the ICs. As Figure 2-6 shows, the temperature rise in the ICs is approximately 20 K to 35 K smaller compared to the natural-convection case due to the higher average fluid velocity. In the forced-convection case, the temperature difference along the board is also less pronounced than for the natural convection case.

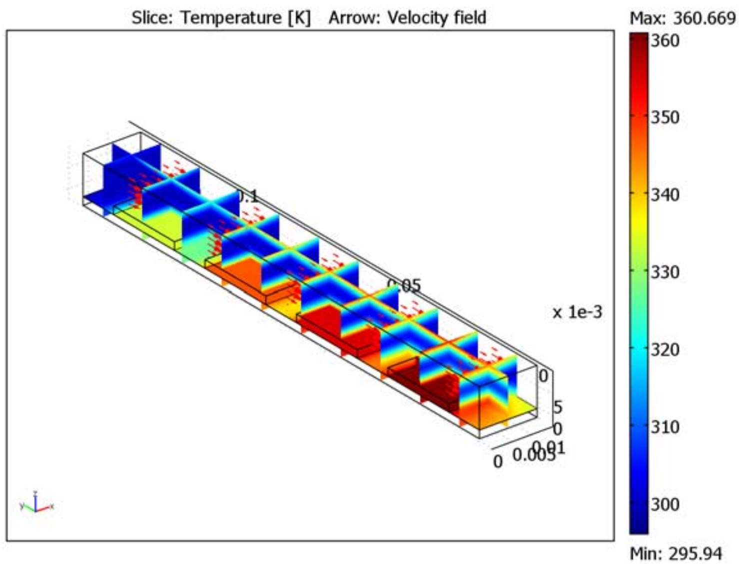


Figure 2-6: Temperature distribution in the case of forced convection cooling of horizontal boards.

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

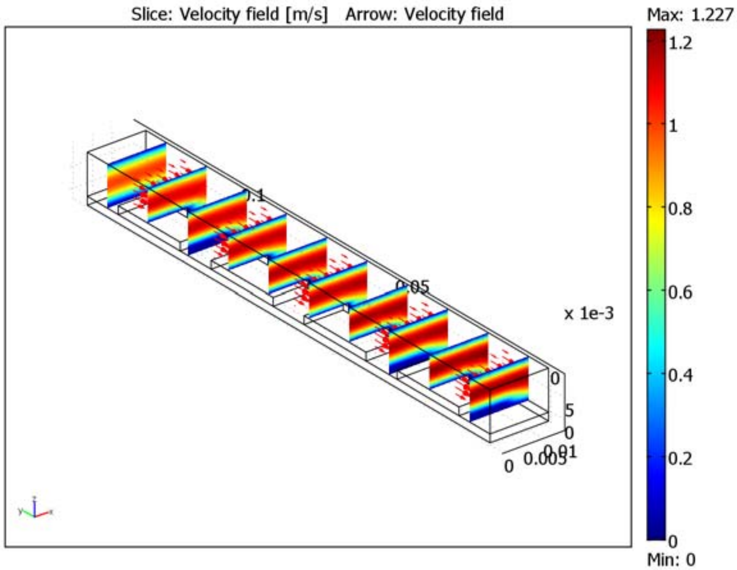


Figure 2-7: Velocity distribution for the case of forced convection.

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 this reads

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

where Ω_1 is the source surface, 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, respectively. 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}. Its definition is similar to h except it uses T_{cup} instead of $T_{\mathrm{f,0}}$. T_{cup} is the cross-section average of the fluid temperature, T_{f}, upstream of each source, defined as

$$T_{cup} = \frac{\int(\rho\mathbf{n} \cdot u)T_f d\Omega_2}{\int(\rho\mathbf{n} \cdot u)d\Omega_2}$$

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

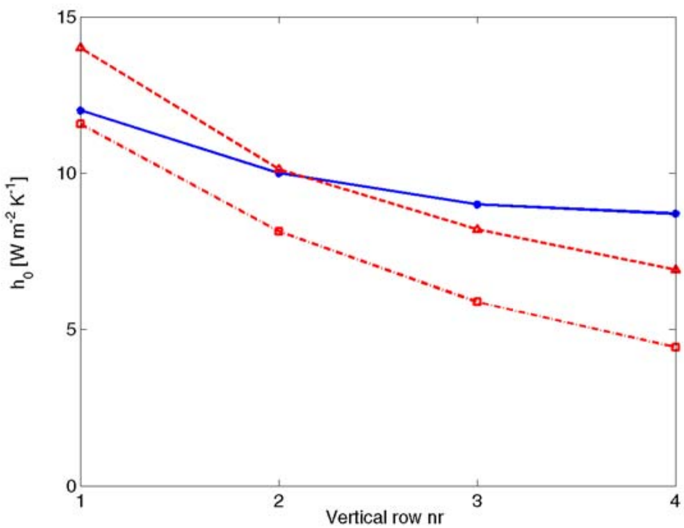


Figure 2-8: Comparison of experimentally measured film resistances, h (asterisks, solid line), for natural convection (Ref. 1) with those calculated from the 2D model (squares, dash-dot line) and the 3D model (triangles, dashed line).

The deviation from the experimental values for the natural convection might stem from differences in the geometry (which is not fully defined in Ref. 1).

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

$$\mathrm{Nu}_L = h_{ad}(L/k)$$

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 (Ref. 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 and simulated values indicates the models' high accuracy.

References

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

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

3. J.M. Coulson and J.F. Richardsson, *Chemical Engineering*, Vol. 1, Pergamon Press, 1990, appendix.

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 COMSOL Multiphysics, and in the **Model Navigator** click the **New** tab.
2. From the **Space dimension** list, select **2D**.
3. From the **Application Modes** tree, select **Heat Transfer Module>Fluid-Thermal Interaction>Non-Isothermal Flow**.
4. Click **OK**.

OPTIONS AND SETTINGS

1. From the **Options** menu, select **Constants**, and in the resulting dialog box define the following names and expressions. When finished, click **OK**.

NAME	EXPRESSION
q_source	(2/3)*1[W]/(20*20*2[mm^3])
T0	300[K]

NAME	EXPRESSION
rho0_air	1.013e5[Pa]*28.8[g/mol]/(8.314[J/(mol*K)]*T0)
Cp_air	1.1[kJ/(kg*K)]

2 From the **Options** menu, select **Expressions>Scalar Expressions**, then define the following names and expressions; when finished, click **OK**.

NAME	EXPRESSION
k_air	10^(-3.723+0.865*log10(abs(T[1/K])))[W/(m*K)]
rho_air	1.013e5[Pa]*28.8[g/mol]/(8.314[J/(mol*K)]*T)
eta_air	6e-6[Pa*s]+4e-8[Pa*s/K]*T

GEOMETRY MODELING

1 Create three rectangles. To do so, go to the **Draw** menu, select **Specify Objects>Rectangle**, and then enter the information from the following table; after creating each rectangle, click **OK**.

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** button on the Main toolbar.

3 In the drawing area, select the rectangle designated R3, then click the **Array** button on the Draw toolbar. In the resulting dialog box, go to the **Displacement** area and in the **y** edit field type 0.03; go to the **Array size** area and in the **y** edit field type 4. Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 In the **Multiphysics** menu, select the **Weakly Compressible Navier-Stokes (chns)** application mode.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomains 1 and 3–6. Select **Solid domain** from the **Group** list. This deactivates the **Weakly Compressible Navier-Stokes** application mode in these subdomains.

4 Select Subdomain 2 and enter these expressions in the appropriate edit fields:

PARAMETER	EXPRESSION
η	eta_air
F_y	9.81[m/s^2]*(rho0_air-rho_htgh)

The density will automatically be imported from the General Heat Transfer application mode.

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

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

7 From the **Physics** menu, open the **Subdomain Settings** dialog box.

8 Click the **General** tab and select Subdomain 1.

9 Select **Solid domain** from the **Group** list and enter the following expressions:

SETTINGS	SUBDOMAIN 1
k	0.3
ρ	1900
C_p	1369

10 Select Subdomain 2 and enter `k_air` in the **k** edit field, `rho_air` in the ρ edit field, and `Cp_air` in the $\mathbf{C_p}$ edit field.

11 Click the **Convection** tab. Select the **Enable convective heat transfer** check box, then select **Ideal gas** from the **Matter state** list.

12 Click the **Ideal Gas** tab.

13 Click the $\mathbf{M_n}$ option button and enter `0.0288` in the corresponding edit field.

14 Click the **General** tab and select Subdomains 3–6.

15 Select **Solid domain** from the **Group** list.

16 Click the **Load** button to open the **Materials/Coefficients Library**.

17 From the **Basic Material Properties** library, choose **Silicon**, then click **OK**.

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

19 Select all subdomains. Go to the **Init** page, and in the **Temperature** edit field enter `T0`.

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

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Set the boundary conditions as in the following table; when done, click **OK**.

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

3 In the **Multiphysics** menu, select **Weakly Compressible Navier-Stokes (chns)**.

4 From the **Physics** menu, open the **Boundary Settings** dialog box.

You must only change the boundary condition at in- and outlet because no slip is default for all boundaries.

5 Select Boundaries 5 and 22, then set the **Boundary type** to **Open boundary** and the **Boundary condition** to **Normal stress**. Leave $\mathbf{f_0}$ at zero.

6 Click **OK**.

7 From the **Physics>Periodic Conditions** menu, open the **Periodic Boundary Conditions** dialog box.

8 On the **Source** page select Boundary 1, go to the **Expression** edit field and type `T`, then press Enter.

9 Click the **Destination** tab, select the check box corresponding to Boundary 27, and in the **Expression** edit field type `T`.

10 Click the **Source Vertices** tab, find the **Vertex selection** list, select Vertices 1 and 2, then click the **>>** button.

11 Click the **Destination Vertices** tab. Select and add Vertices 21 and 22. Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box. Go to the **Global** page. From the **Predefined mesh sizes** list select **Normal**.

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

3 On the **Mesh** menu, select **Initialize Mesh**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar 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 Click the **Surface** tab.
3 On the **Surface Data** page, select **General Heat Transfer (htgh)>Temperature** from the **Predefined quantities** list. Click **Apply**.
4 To reproduce Figure 2-5 (a) select **Weakly Compressible Navier-Stokes (chns)>Velocity field** from the **Predefined quantities** list on the **Surface Data** page, then 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`

Repeat the steps from the 2D model in the sections "Model Navigator" and "Options and Settings" with two exceptions: in the **Space dimension** list select **3D** and `q_source` should be equal to `1[W]/(20*20*2[mm^3])`.

GEOMETRY MODELING

1 Create three blocks using data from this table. To do so, go to the **Draw** menu and select **Block**.

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** button on the Main toolbar.
3 Select the object BLK2, then click the **Array** button on the Draw toolbar. Go to the **Displacement** area and in the **z** edit field type `0.03`. Go to the **Array size** area and in the **z** edit field type `4`. Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 From the **Multiphysics** menu, select the **Weakly Compressible Navier-Stokes (chns)** application mode.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomains 2–6. Select **Solid domain** from the **Group** list. This will deactivate the Weakly Compressible Navier-Stokes application mode in these subdomains.

4 Select Subdomain 1, then enter the following expressions in the appropriate edit fields:

PARAMETER	EXPRESSION
η	`eta_air`
F_z	`9.81[m/s^2]*(rho0_air-rho_htgh)`

The density will automatically be imported from the General Heat Transfer application mode.

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

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

7 From the **Physics** menu, open the **Subdomain Settings** dialog box.

8 Click the **General** tab and select Subdomain 6.

9 From the **Group** list, select **Solid domain**. Then enter the following settings:

SETTINGS	SUBDOMAIN 6
k (isotropic)	0.3
ρ	1900
C_p	1369

10 Select Subdomain 1. Enter `k_air` in the **k** edit field and `Cp_air` in the $\mathbf{C_p}$ edit field.

11 Click the **Convection** tab. Select the **Enable convective heat transfer** check box, then select **Ideal gas** from the **Matter State** list.

12 Click the **Ideal gas** tab.

13 Click the $\mathbf{M_n}$ button and enter `0.0288` in the corresponding edit field.

14 Click the **General** tab, then select Subdomains 2–5.

15 From the **Group** list, select **Solid domain**.

16 Click the **Load** button to open the **Materials/Coefficients Library**.

17 From the **Basic Material Properties** library, choose **Silicon**, then click **OK**.

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

19 Select all subdomains. Click the **Init** tab, then in the **Temperature** edit field type `T0`.

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

Boundary Conditions

1 From the **Physics** menu, select **Boundary Settings**.

2 Set the boundary conditions as follows; when done, click **OK**.

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

3 From the **Multiphysics** menu, select **Weakly Compressible Navier-Stokes (chns)**.

4 From the **Physics** menu, select **Boundary Settings**.

The default boundary condition is no-slip. Hence, only those boundaries that are not walls have to be specified.

5 Select Boundaries 3 and 4, then set the **Boundary type** to **Open Boundary** and the **Boundary Condition** to **Normal stress**. Leave f_0 at zero.

6 Select Boundaries 1 and 34, then apply the boundary Type **Symmetry Boundary**.

7 Click **OK**.

8 From the **Physics** menu, select **Periodic Conditions>Periodic Boundary Conditions**.

9 On the **Source** page select Boundary 2. In the **Expression** edit field type `T`, then press Enter.

10 Click the **Destination** tab, and click the check box to select Boundary 29. In the **Expression** field type `T`.

11 Click the **Source Vertices** tab. Select and add (using the **>>** button), in this order, Vertices 1, 2, 39, and 40.

12 Click the **Destination Vertices** tab. Select and add Vertices 21, 22, 43, and 44, again in the mentioned order. Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, select **Free Mesh Parameters**. On the **Global** page, go to the **Predefined mesh sizes** list and select **Finer**.

2 Click the **Boundary** tab and select Boundary 2. In the **Maximum element size** edit field, type `2e-3`.

3 Click the **Advanced** tab. In the **x-direction scale factor** edit field type `0.5`. Click **OK**.

4 On the **Mesh** menu, select **Initialize Mesh**.

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Parameters** dialog box.

2 On the **General** page, select **Direct (PARDISO)** from the **Linear system solver** list.

3 Click **OK**.

4 Click the **Solve** button on the Main toolbar to compute the solution.

The problem is rather large because of the strong coupling between temperature and velocity and because of the dense mesh. A computer needs approximately 500 MB of free memory to solve the problem. The problems can take a few minutes to solve.

POSTPROCESSING AND VISUALIZATION

To generate the temperature plot in Figure 2-4 execute the following instructions:

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2 On the **General** page, clear the check box for the **Boundary** plot type and select the check box for the **Slice** plot type.

3 Click the **Slice** tab. Go to the **Slice data** area. In the **Predefined quantities** list, select **General Heat Transfer (htgh)>Temperature**.

4 Go to the **Slice positioning** area. In the **Number of levels** edit fields for **x levels** and **z levels** type `1` and `8`, respectively. For **y levels**, select the option button next to the **Vector with coordinates** edit field, then type `0` in this edit field.

5 Click **Apply** to launch the plot.

To generate the plot in Figure 2-5 (b), proceed as follows:

6 Still on the **Slice** page, select **Weakly Compressible Navier-Stokes (chns)>Velocity field** from the **Predefined quantities** list.

7 Click the option button next to the **Number of levels** edit field for **y levels**, then type `0` in this edit field. Click **Apply**.

Finally, you can reproduce the model image—that is, the plot shown in the **Model Navigator** and when the model opens—by the following modifications:

8 In the **Predefined quantities** list on the **Slice** page, select **General Heat Transfer (htgh)>Temperature**.

9 Click the **Arrow** tab. Select the **Arrow plot** check box.

10 In the **Predefined quantities** list on the **Subdomain Data** page, leave the default selection, which is **Weakly Compressible Navier-Stokes (chns)>Velocity field**.

11 Go to the **Arrow positioning** area. In the **Number of points** edit fields for **x points**, **y points**, and **z points** type `5`, `5`, and `8`, respectively.

12 In the **Arrow parameters** area, select **3D arrow** from the **Arrow type** list, then 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`

You implement this model by modifying the previous one (circuit_board_nat_3D.mph). Begin by loading or building that previous model.

GEOMETRY MODELING

1 Click the **Draw Mode** button on the Main toolbar to enter the Draw mode.

2 Select all objects, then select the menu item **Draw>Modify>Rotate**.

3 In the **Rotation angle** edit field type `-90`.

4 Go the **Rotation axis direction vector** area, and in the in the **x**, **y**, and **z** edit fields type `1`, `0`, and `0`, respectively. Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

Since you rotated the model geometry, and you are modeling forced convection here, the volume force that acts on the fluid will have different magnitude and direction. The volume force is represented by gravitation. However, for this problem we will neglect the gravitational force, which is a fair assumption. Therefore, zero-out the volume force present in the Weakly Compressible Navier-Stokes application mode according to the following steps:

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select Subdomain 2 and type `0` in the **Volume force, z dir.** edit field.

3 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, select **Boundary Settings**.
2 Select Boundary 29. Set the **Boundary type** to **Inlet** and the **Boundary condition** to **Velocity**.
3 Click the $\mathbf{u_0}$, $\mathbf{v_0}$, $\mathbf{w_0}$ option button.
4 In the **y-velocity** edit field, type `4e4*z*(0.01-z)[1/m^2]*(-1[m/s])`, then click **OK**.
5 From the **Multiphysics** menu, select **General Heat Transfer**.
6 From the **Physics** menu, select **Boundary Settings**.
7 In the dialog box that opens, set the boundary conditions as follows; when done, click **OK**.

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

MESH GENERATION

1 From the **Mesh** menu, select **Free Mesh Parameters**. On the **Global** page verify that **Finer** is selected from the **Predefined mesh sizes** list.
2 Click the **Custom mesh size** button and set the **Resolution of narrow regions** parameter to `1`.
3 On the **Boundary** page, select Boundary 7. In the **Maximum element size** edit field type `2.8e-3`, then click **OK**.
4 Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Manager** dialog box.
2 On the **Initial Value** page, go to the **Initial value** area and select the **Initial value expression** option button.
3 Click **OK**.
4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 To create Figure 2-6, first go to the **Postprocessing** menu and open the **Plot Parameters** dialog box.

2 On the **General** page, select the check box for the **Slice** plot type and clear the check box for the **Boundary** plot type.

3 Click the **Slice** tab, then go to the **Slice data** area. In the **Predefined quantities** list select **General Heat Transfer (htgh)>Temperature**.

4 Go to the **Slice positioning** area. In the **Number of levels** edit fields for **x levels** and **y levels** type 1 and 10, respectively. For **z levels**, click the option button next to the **Vector with coordinates** edit field, then type 0 in this edit field. Click **Apply**.

5 To produce Figure 2-7, select **Weakly Compressible Navier-Stokes (chns)>Velocity field** from the **Predefined quantities** list.

6 In the **Number of levels** edit field for **x levels** type 0. Click the option button next to the **Number of levels** edit field for **z levels**, then type 0 in this edit field. Click **OK**.

Generate the plot shown in the **Model Navigator** and when the model opens in the following way:

1 From the **Postprocessing** menu, select **Plot Parameters**.

2 On the **General** page clear the check boxes for the **Slice** and **Arrow** plot types and then select the check boxes for the **Boundary** and **Streamline** plot types.

3 Click the **Streamline** tab.

4 On the **Start Points** tab click the **Specify start point coordinates** button.

5 In the **x** edit field type `linspace(1e-3,13e-3,13)`.

6 In the **y** edit field type `linspace(0.13,0.13,13)`.

7 In the **z** edit field type `linspace(1.3e-3,1.3e-3,13)`.

8 Click the **Line Color** tab, then click the **Use expression** button.

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

10 In the **Expression** field type `T-296[K]`, then click **OK**.

11 In the **Line type** list, select **Tube**.

12 Click the **Tube Radius** button at the bottom of the dialog box.

13 Click the **Radius data** check box, and in the **Predefined quantities** list select **General Heat Transfer (htgh)>Temperature gradient**.

14 Clear the **Radius scale factor>Auto** check box, and in the edit field for the scale factor enter `0.5`. Click **OK**.

15 Click the **Advanced** button. In the **Maximum number of integration steps** edit field type `1000`, then click **OK**.

16 On the **Boundary page**, select **General Heat Transfer (htgh)>Temperature** from the **Predefined quantities** list.

17 Click **OK**.

18 From the **Options** menu, select **Suppress>Suppress Boundaries**.

19 Select Boundaries 4–7, 9, 29, and 35, then click **OK**.

20 Click the **Postprocessing Mode** button on the Main toolbar.

21 Click the **Scene Light** button on the Camera toolbar.

22 Double-click the **AXIS** button on the status bar at the bottom of the COMSOL Multiphysics window to disable the coordinate axes.

Forced Convection Cooling—Simplified Models

The two following examples illustrate simplified approaches to simulating forced convection cooling. This discussion starts with the exact problem from the section "Forced Convection—Horizontal Boards" on page 17 and shows how to simplify it. Specifically, if you know the heat transfer film coefficient, h, it is not necessary to include the flow field; the General Heat Transfer application mode is then sufficient for modeling the temperature distribution. And while the following examples take their h values from the results of the rigorous model "Convection Cooling of Circuit Boards" on page 10 (the case of forced convection), you can also use the methodology of known values or expressions for h.

The dimensions of the problem geometry (Figure 2-9) and its parameters are the same as in the previous example. In brief, the system cools a stack of circuit boards with four in-line ICs, each producing 1 W of heat, through forced convection. The aim of both of the following models is to determine the temperature development of the board and ICs.

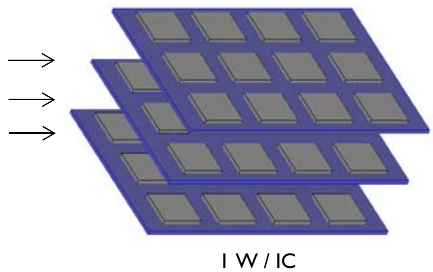


Figure 2-9: Starting geometry for the problem.

1D PLUG FLOW

First, this example sets up a 1D adiabatic plug-flow model describing the cup mixing temperature of the air (fluid), $T_{\mathrm{f,cup}}$, in the channel between the boards during forced convection. It uses the equation

$$T_{\mathrm{f,cup}} = \frac{\int(\rho\mathbf{n}\cdot\mathbf{u})T_{\mathrm{f}}d\Omega_2}{\int(\rho\mathbf{n}\cdot\mathbf{u})d\Omega_2}.$$

The model does not include the temperature distribution in the air. In addition, the model assumes the sources are infinite in the board's lateral direction. Thus, in principle the model describes the distribution of temperature in the flow direction along a line in the air channel. Figure 2-10 depicts the resulting 1D geometry.

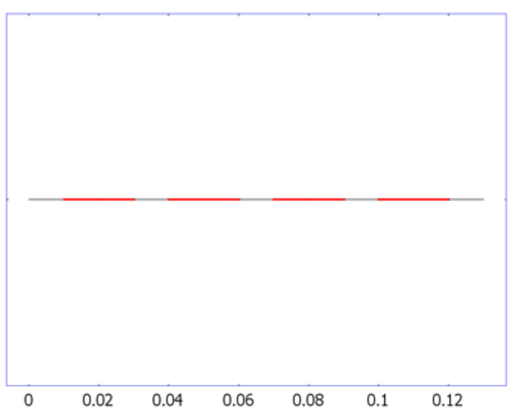


Figure 2-10: The geometry of the 1D-model.

The model uses the General Heat Transfer application mode. It sets the convective velocity to 0.667 m/s at the inlet (that is, the average velocity of the previous models) and assume that it varies with temperature according to

$$u(x) = u_0 \frac{T_{\mathrm{f,cup}}}{T_0}.$$

The next equation describes the heat transfer

$$\nabla \cdot (-k \nabla T_{\mathrm{f,cup}}) = Q - \rho C_p \mathbf{u} \cdot \nabla T_{\mathrm{f,cup}}$$

where k represents the thermal conductivity, C_p gives the specific heat capacity, and Q is the heating power per unit volume. The model sets Q to zero for the subdomains between the sources while it equals 1666.67 W/m^2 (that is, $(2/3)\cdot 1$ W/ $((20\cdot 10^{-3})^2$ m^2)) at the source subdomains. The factor $2/3$ represents the lateral average heating power, taking the open slots between the ICs into account.

The material properties are the same as those in the previous models, also taking temperature variations into account. At the inlet boundary, the temperature is fixed to 300 K, and at the outlet the model applies convective heat flux.

The goal is to calculate the ICs' surface temperature, T_{s}. It is a function of the fluid temperature and the adiabatic heat transfer film coefficients, h_{ad}, according to

$$T_{\mathrm{s}} = \frac{q}{h_{\mathrm{ad}}} + T_{\mathrm{f,cup}}$$

where q is the heat flux. This equation calculates the IC surface temperature.

This example calculates values of h_{ad} using the results of the previous 3D model ("Forced Convection—Horizontal Boards" on page 17) with the formula

$$h_{\mathrm{ad}} = \frac{Q_{\mathrm{tot}}}{A_{\mathrm{2D}}(T_S - T_{\mathrm{f,cup}})}$$

where A_{2D} is the IC's xy-projected area, and Q_{tot} is the IC's total heating power (in this case, 1 W).

You can easily perform these calculations using the postprocessing capabilities of COMSOL Multiphysics. Specifically, with the model "circuit_board_3d_forced" open, first calculate $T_{\mathrm{f,cup}}$ for each cross section of the air domain. Next calculate the total heat flux, Q_{tot}, from each surface. Finally use the equation just given to derive the following values of h_{ad} (a discussion of h^0 follows shortly):

REGION	h_{ad} (W/(m^2·K))	h^0 (W/(m^2·K))
Source 1	57.5	35.0
Source 2	44.4	22.1
Source 3	40.6	19.2
Source 4	39.2	18.0

SIMPLIFIED 3D MODEL

This second example sets up a transient 3D model describing the temperature of the board and ICs during startup. In this case the modeled geometry consists of the board and ICs but not the air. The simplified model makes it possible to investigate the temperature transient of a entire row of ICs.

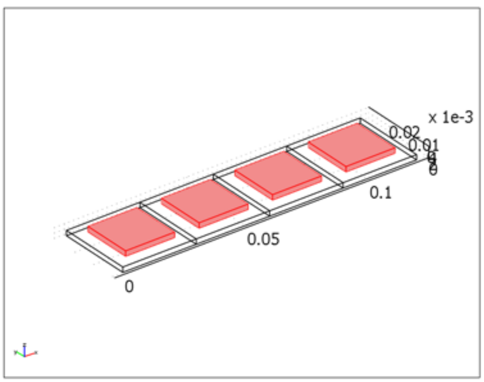


Figure 2-11: Geometry of the 3D model.

In contrast to the model in the previous section, which used both conduction and convection, this model works only with the conduction feature of the General Heat Transfer application mode. It uses the isothermal film coefficients, h^0, to calculate the convective cooling. You calculate them from the results of the previous 3D model (“Forced Convection—Horizontal Boards” on page 17), doing so in a similar way as you did for the 1D-plug-flow model just described except using the formula

$$h^0 = \frac{Q_{tot}}{A_{2D}(T_S - T_0)}$$

where T_0 is the air's inlet temperature. To model the heat transfer coefficient of the board, a function from the built-in Heat Transfer Coefficients library is used. The function is valid for forced convection on plates. For more information about the Heat Transfer Coefficients library, see the *Heat Transfer Module User's Guide*.

Further, the material properties specified in the subdomain settings for this model are identical to those in the previous models. The initial temperature of all components is 300 K, as is the surrounding temperature. For the ICs it applies a volume heat source of 1.25 MW/m. In the heat flux boundary conditions, for the downside segments of the board and for the circuit surface boundaries, it uses the h^0 values.

Results and Discussion

1D PLUG FLOW

Figure 2-12 shows the results of the 1D model for the ICs' surface temperature.

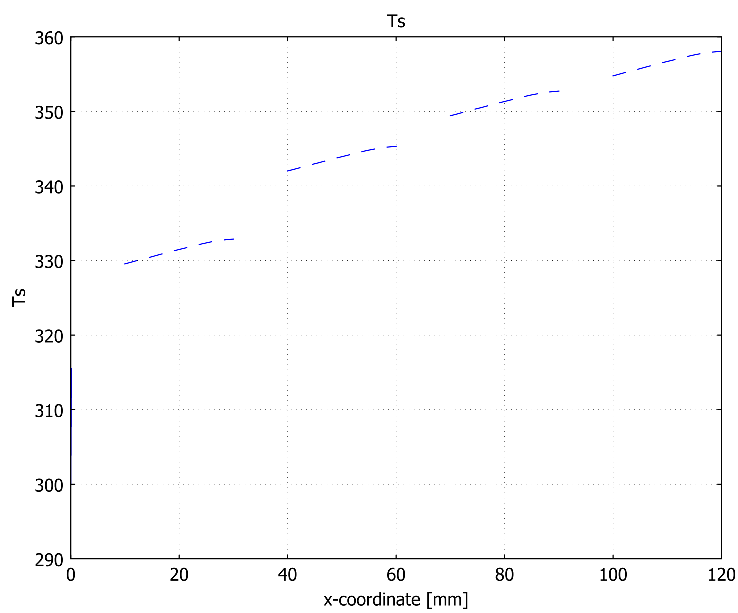


Figure 2-12: 1D model results for the surface temperature of the ICs (dashed line) and the average temperature of the fluid (solid line).

The profile agrees rather well with that of the previous 3D model, in this case experiencing a maximum surface temperature of 357 K. This indicates that you can model the heat transfer with good accuracy in a simplified way if you know the values of the film coefficient, h_{ad}. The simplified 1D model is thus a good predictor even though it does not simulate the temperature distribution in the fluid and the fluid flow field.

SIMPLIFIED 3D MODEL

This model results in an accurate determination of the source surface temperatures. A benefit of having an easy-to-solve model is that you can proceed and analyse the transient behavior. Figure 2-13 shows the transient 3D model results at 1000 s.

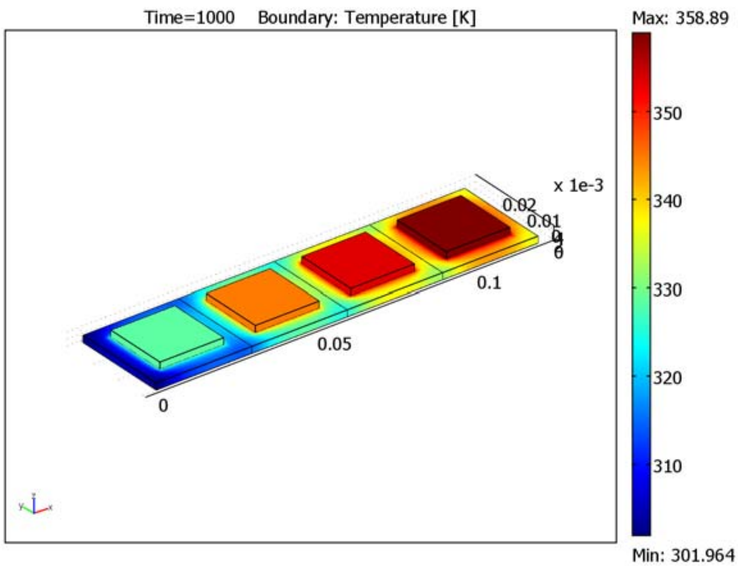


Figure 2-13: Temperature of the source surfaces 1000 s after applying the heat load according to the simplified 3D model.

The results indicate that this amount of time is approximately sufficient to reach steady state.

Modeling Using the Graphical User Interface—1D Plug Flow

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/simplified_circuit_board_1d`

MODEL NAVIGATOR

1 In the **Model Navigator** click the **New** tab, and in the **Space dimension** list select **1D**.

2 From the **Application Modes** tree, select **Heat Transfer Module>General Heat Transfer**.

3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, select **Constants**. In the dialog box, define the following names and expressions; when done, click **OK**.

NAME	EXPRESSION
v_in	(2/3)*1[m/s]
T0	300[K]
width	0.01
q_s	(2/3)*1.25[MW/m^3]*2[mm]
h1	57.5[W/(m*K)]
h2	44.4[W/(m*K)]
h3	40.6[W/(m*K)]
h4	39.2[W/(m*K)]

2 From the **Options** menu, select **Expressions>Scalar Expressions**. In the dialog box, define the following names and expressions; when done, click **OK**.

NAME	EXPRESSION
k_f	10^(-3.723+0.865*log10(T[1/K]))[W/(m*K)]
rho_f	1.013e5[Pa]*28.8[g/mol]/8.314[J/(mol*K)]/T
Cp_f	1.1[kJ/(kg*K)]
v	v_in*width*T/T0

GEOMETRY MODELING

1 Create three line segments. To do so, go to the **Draw** menu and select **Specify Objects>Line**.

2 In the **Line** dialog box, enter the following settings; when done, click **OK**.

LINE SEGMENT	X-COORDINATES
1	`0 0.01`
2	`0.01 0.03`
3	`0.03 0.04`

3 Click the **Zoom Extents** button on the Main toolbar.

4 Copy line segments 2 and 3 by selecting these objects and pressing Ctrl+C.

5 To complete the geometry, perform a paste operation three times by pressing Ctrl+V; each time use a different **Displacement** in the **x** edit field of `0.03`, `0.06`, and `0.09`.

6 Once again click the **Zoom Extents** button.

PHYSICS SETTINGS

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 On the **Init** page, select all subdomains, then in the **T(t_0)** edit field type `T0`.

3 On the **General** page, enter these settings:

SETTINGS	SUBDOMAINS 1, 3, 5, 7, 9	SUBDOMAINS 2, 4, 6, 8
k (isotropic)	`k_f`	`k_f`
ρ	`rho_f`	`rho_f`
C_p	`Cp_f`	`Cp_f`
Q	`0`	`q_s`

4 Click the **Convection** tab, select all subdomains, and click the **Enable convective heat transfer** check box. In the **u** edit field for the *x*-velocity, type `v`. Click **OK**.

5 From the **Options** menu, select **Expressions>Subdomain Expressions**.

6 Enter the name of the subdomain expression and the expressions that define it in the various subdomains; when done, click **OK**.

NAME	SUBDOMAIN 2	SUBDOMAIN 4	SUBDOMAIN 6	SUBDOMAIN 8
`Ts`	`q_s/h1+T`	`q_s/h2+T`	`q_s/h3+T`	`q_s/h4+T`

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Specify the boundary conditions as follows; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARY 10
Boundary condition	Temperature	Convective flux
T_0	`T0`	

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2 Click the **Line** tab.

3 In the **Expression** edit field, type `Ts`, then click **OK**.

4 To reproduce Figure 2-12, go to the **Postprocessing** menu and select **Domain Plot Parameters**.

5 On the **General** page, select the **Keep current plot** check box.

6 On the **Line/Extrusion** page, select all the subdomains, then click **Apply**.

7 In the **y-axis data** area, type `Ts` in the **Expression** edit field.

8 In the **x-axis data** area, click the **Expression** button. In the **X-Axis Data** dialog box, type `x` in the **Expression** edit field. From the **Unit** list, select **mm**.

9 Click **OK** to close the **X-Axis Data** dialog box.

10 Click the **Line Settings** button and from the **Line style** list select **Dashed line**. Click **OK**.

11 Click **OK** to generate the figure.

Modeling Using the Graphical User Interface—3D Model

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/simplified_circuit_board_3d_hcoeff`

MODEL NAVIGATOR

1 Open the **Model Navigator**, click the **New** tab, and from the **Space dimension** list select **3D**.

2 From the **Application Modes** tree, select **Heat Transfer Module>General Heat Transfer**, then click **OK**.

OPTIONS AND SETTINGS

In the **Options** menu, select **Constants**. In the dialog box, define the following names and expressions; when done, click **OK**.

NAME	EXPRESSION
T0	300[K]
q_s	1[W]/(20*20*2[mm^3])
hs1	35.0[W/(m^2*K)]
hs2	22.1[W/(m^2*K)]
hs3	19.2[W/(m^2*K)]
hs4	18.0[W/(m^2*K)]
u_in	1[m/s]

GEOMETRY MODELING

1 Create two blocks according to the specifications in the table below. To do so, select **Block** from the **Draw** menu; click **OK** after specifying each block.

OBJECT	LENGTH X	LENGTH Y	LENGTH Z	BASE X	BASE Y	BASE Z
BLK1	0.03	0.03	0.002	0	0	0
BLK2	0.02	0.02	0.002	0.005	0.005	0.002

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

3 Select both blocks with the mouse, then go to the **Draw** menu and select **Modify>Array**.

4 In the **Displacement** area, go to the **x** edit field and type 0.03; in the **Array size** area find the **x** edit field and type 4. Click **OK**.

PHYSICS SETTINGS

Subdomain Settings

1 From the **Physics** menu, open the **Subdomain Settings** dialog box.

2 On the **Init** page, select all subdomains, then in the **T(t_0)** edit field type T0.

3 Click the **General** tab and enter the following settings. For Subdomains 2, 4, 6, and 8, click the **Load** button and select **Silicon** from the **Basic Material Properties** library

in the **Materials/Coefficients Library** dialog box. This defines k, ρ, and C_p for those subdomains. When done, click **OK**.

QUANTITY	SUBDOMAINS 1, 3, 5, 7	SUBDOMAINS 2, 4, 6, 8
k (isotropic)	0.3	163
ρ	1900	2330
C_p	1369	703
Q	0	q_s

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select Boundaries 3, 4, 14, 15, 25, 26, 36, and 37. Specify a **Heat flux** boundary condition.

3 Click the **Load** button to load a heat transfer coefficient. This opens the **Materials/Coefficients Library** dialog box.

4 In the **Materials/Coefficients Library** dialog box, select **Heat Transfer Coefficients>Air, Ext. Forced Convection>Forc. Plate, h_loc, s=position, U=velocity**, then click **OK**. This brings you back to the **Boundary Settings** dialog.

5 Edit the function call expression in the **h** edit field to read `h_loc(T,Tinf_htgh,x,u_in)`.

6 Type `T0` in the **T_{inf}** edit field for the external temperature.

7 Set the remaining boundary conditions as follows; when done, click **OK**.

SETTINGS	BOUNDARIES 6, 7, 9–11	BOUNDARIES 17, 18, 20–22	BOUNDARIES 28, 29, 31–33	BOUNDARIES 39, 40, 42–44
Boundary condition	Heat flux	Heat flux	Heat flux	Heat flux
h	hs1	hs2	hs3	hs4
T_{inf}	T0	T0	T0	T0

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Parameters** dialog box.

2 In the **Solver** list, select **Time dependent**. In the **Times** edit field, type `0 1000`.

3 Click the **Time Stepping** tab. In the **Times to store in output** list, select **Time steps from solver**. Click **OK**.

4 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

1 To generate an animation of temperature over time (and reproduce Figure 2-13), go to the **Postprocessing** menu and open the **Plot Parameters** dialog box.

2 On the **General** page, go to the **Plot type** area and clear the check box next to **Slice**, then select the check box next to **Boundary**.

3 On the **Animate** page click the **Start Animation** button at the bottom of the dialog box. The software now generates the animation, which might take a few seconds. To replay the animation, use the icons in the lower left corner of the **COMSOL Movie Player** dialog box.

Forced Turbulent Convection Cooling

Introduction

The following set of models demonstrates how to model a conjugate heat transfer problem with COMSOL Multiphysics. The models show two different approaches. The first one uses the Turbulent Non-Isothermal Flow predefined multiphysics coupling from the Heat Transfer Module. The second approach is a simplified one making use of the Heat Transfer Coefficients library supplied with the Heat Transfer Module. In addition, this discussion shows how to modify the k-ε Turbulence Model application mode's equations to take density variations into account (weakly compressible flow).

Figure 2-14 depicts the geometry: a horizontal stream of air that cools a thin and infinitely wide horizontal plate. The plate is at a uniform temperature at the bottom, and the flow is turbulent. This is a well-studied case of convection cooling that works well as a benchmark that demonstrates the accuracy of the modeling methods.

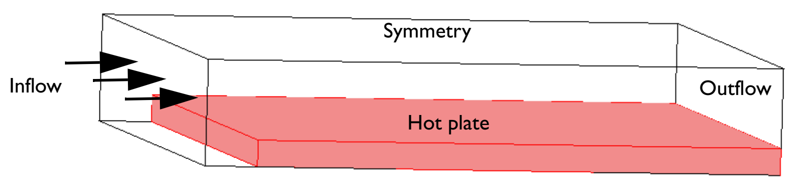


Figure 2-14: Forced convection cooling of a horizontal plate.

You can take two main approaches when simulating forced convection cooling: first, model the heat transfer by using heat transfer coefficients; second, also solve for the fluid flow field and include heat transfer in the fluid domain. The first approach works well for simple geometries such as the one in this example, for which accurate heat transfer coefficient expressions and correlations exist. For more complex geometries, however, such correlations might not describe the situation very well, and so the second approach is the best choice. If you are interested in the flow field or the temperature distribution in the fluid, the second alternative is, of course, the only choice. This exercise explains how to set up both approaches and then compares the results.

Model Definition

SOLID AND FLUID HEAT TRANSFER—INCLUDING THE FLUID DYNAMICS

The model works with the following equations:

- Reynolds-averaged Navier-Stokes (RANS) equations in the air domain.
- The conductive and convective heat equation in the air and the solid (copper) wall.

The Turbulent Non-Isothermal Flow predefined multiphysics coupling sets up these application modes together with applicable couplings, making it easy to model the fluid-thermal interaction.

The material properties for the fluid are those of air at atmospheric pressure, and for the solid plate those of copper. You can load these properties from the built-in materials library where the air properties are temperature dependent.

It is necessary to correct the fluid's thermal conductivity to take into account the effect of mixing due to eddies. The turbulence results in an effective thermal conductivity, k_{eff}, according to the equation

$$k_{\text{eff}} = k + k_T \qquad k_T = \frac{C_p \eta_T}{Pr_T}.$$

Here k is the physical thermal conductivity of the fluid, k_T is the turbulent conductivity, η_T denotes the turbulent viscosity, C_p equals the heat capacity and Pr_T is the turbulent Prandtl number. With COMSOL Multiphysics you can easily obtain the effective conductivity by using the 'default' group setting in the fluid domain. In the group, the variable for turbulent conductivity is already given in the General Heat Transfer application mode for the fluid.

Figure 2-15 depicts the model with its boundary conditions.

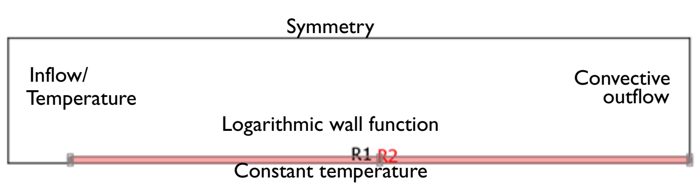


Figure 2-15: Modeled 2D geometry with boundary conditions.

The boundary conditions for the problem are:

- k-ε equations in the fluid domain
  - Specified velocity at the inlet
  - Pressure and no viscous stress at the outlet
  - Symmetry at the top boundary
  - Logarithmic wall function at the plate's surface boundaries
- Heat transport equations
  - Room temperature at the inlet
  - Convection-dominated transport at the outlet
  - Symmetry at the top boundary
  - Thermal wall function at the plate/air interface
  - Fixed temperature at the bottom of the heated plate

To model the solid-fluid interfaces, the model uses the logarithmic wall function boundary condition for turbulent flow, in which an algebraic relationship—the logarithmic wall function—describes the momentum transfer at the solid-fluid interface. This means that the modeled domain ends at the top of the laminar boundary layer where the fluid experiences a significant wall-tangential velocity. This is an important aspect to consider when modeling the heat transfer. Like the fluid velocity, the temperature is not modeled in the laminar sublayer. Instead of assuming the temperature to be continuous across the layer, the model uses a thermal wall function. This creates a jump in temperature between the solid surface and the fluid due to the omitted laminar sublayer. The predefined group for the wall domains defines this wall function in the following way.

To implement the thermal wall function, the model uses two heat transfer application modes: one for the solid and one for the fluid. These are connected through a heat flux boundary condition, the thermal wall function. This means that the resistance to heat transfer through the laminar sublayer is related to that for momentum transfer for the fluid. You therefore determine the heat flux, q, from the equation

$$q = \frac{\rho C_p C_\mu^{1/4} {k_\mathrm{w}}^{1/2} (T_\mathrm{w} - T)}{T^+}$$

where ρ and C_p are the fluid's density and heat capacity, respectively; C_μ is a numerical constant of the turbulence model; and k_w is the value of the turbulent kinematic energy at the wall. Furthermore, T_w equals the temperature of the solid at the wall, while T is the fluid temperature on the other side of the omitted laminar sublayer. The dimensionless quantity $T^+ = T^+(\delta_w^+)$ is the dimensionless temperature and depends on the dimensionless wall offset, δ_w^+.

At the front of the hot plate a stagnation point for the flow develops. Typical for two-equation turbulence models such as the k-ε model is an unphysical production of turbulence at stagnation points. The remedy is to apply a *realizability constraint*, which is a physical constraint on the turbulent viscosity. The realizability constraint makes the simulation less stable and is therefore applied only when necessary.

CONVECTION MODELED AS A BOUNDARY CONDITION

This simplified model uses only an energy-balance condition for the solid wall. The heat transfer at the fluid/solid interface is calculated with established theoretical correlations. This means that it is not necessary to model the fluid domain. The model determines the heat transfer at the fluid-cooled side of the wall using a heat transfer coefficient correlation from the built-in library. If the aspect of primary interest is heat transfer at the wall/fluid interface, then this method is very useful.

The simplified model uses the same geometry although it applies the heat transfer equations only inside the plate. The model works with the heat transport equations in the solid (copper) plate. For this purpose you can use a General Heat Transfer application mode from the Heat Transfer Module.

The boundary conditions for the heat transport equations are

- Fixed temperature at the bottom of the heated plate
- Flux boundary condition at the plate's top boundary (interface with fluid) using a heat transfer coefficient.

To describe the heat transfer coefficient for atmospheric air under various conditions, this example models the heat transfer coefficient using the built-in heat transfer coefficient library, which is based on general Nusselt correlations.

Results for the Flow/Heat Model

The example solves the problem for a set of inlet velocities between 1 m/s and 100 m/s. Figure 2-16 depicts the temperature distribution for the inlet velocity 1 m/s.

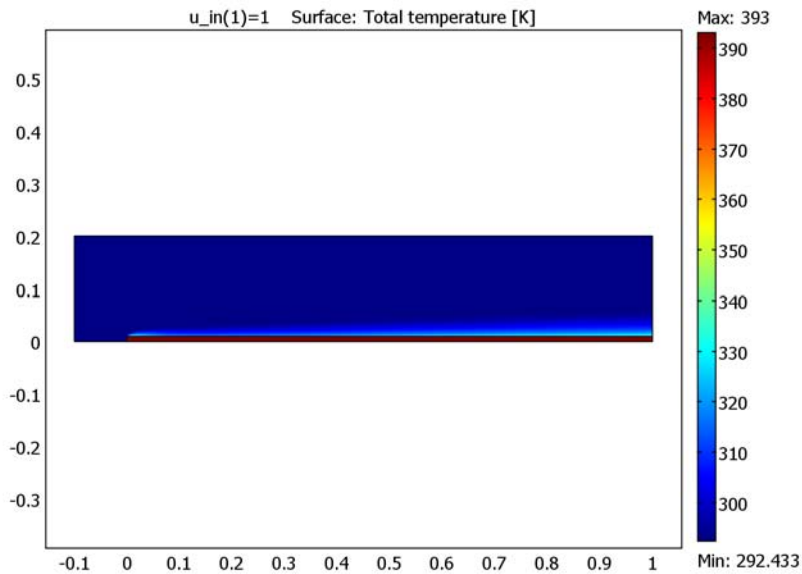


Figure 2-16: Temperature distribution at an inlet velocity of 1 m/s.

The heated layer of air at the plate surface is rather thick considering the relatively high velocity. This is an effect of the turbulent thermal conductivity caused by the eddies in the flow. The next figure depicts the turbulent thermal conductivity of the air.

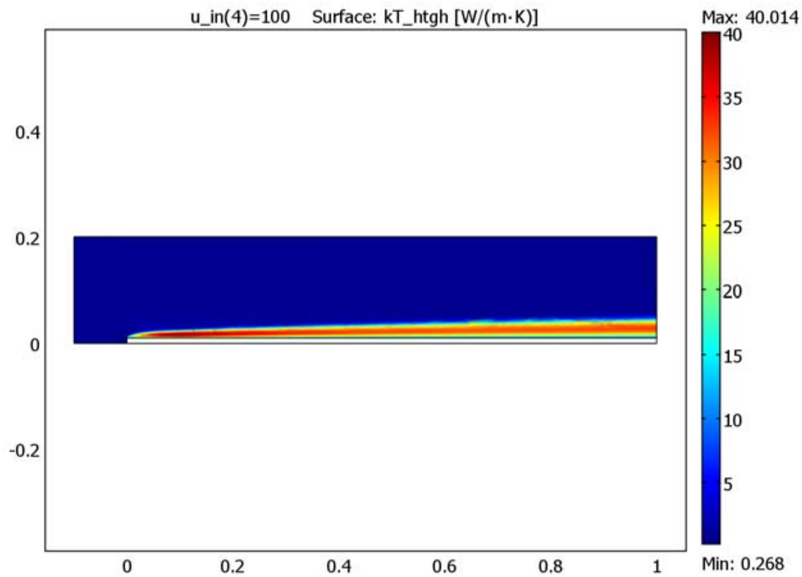


Figure 2-17: Turbulent thermal conductivity of the air at an inlet velocity of 100 m/s.

The turbulent conductivity is much higher than the physical thermal conductivity of air, which is 0.03 W/(m·K) at 323 K. This means that the added turbulent conductivity dominates over the laminar conductivity, and hence that the turbulent eddies cause a significantly higher heat flux at the cooled surface compared to a laminar flow.

In this example you also modify the turbulent flow model to take density variations into account. The density of air decreases with temperature; the following figure shows its variation at an inlet velocity of 1 m/s.

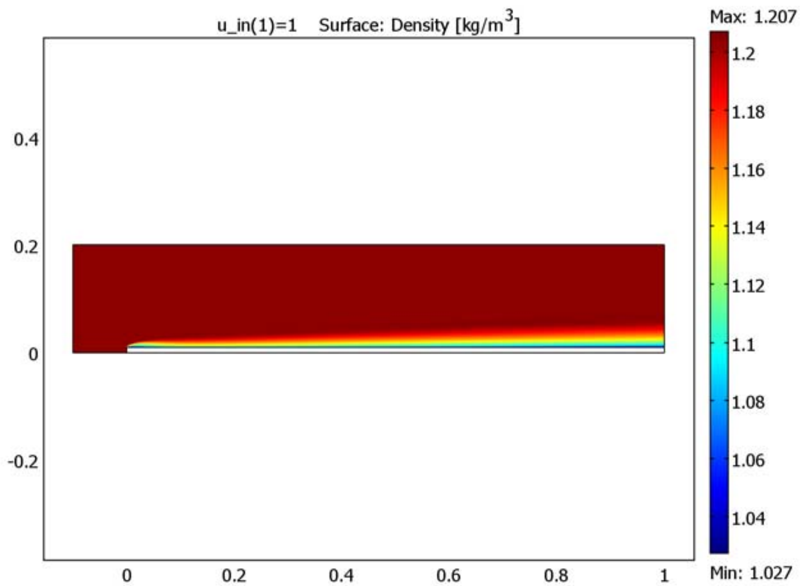


Figure 2-18: Fluid density at an inlet velocity of 1 m/s.

These results point out the importance of taking density variations into account. As the density decreases, the fluid velocity increases. This effect becomes apparent in the next figure, which shows the velocity distribution at the same inlet velocity.

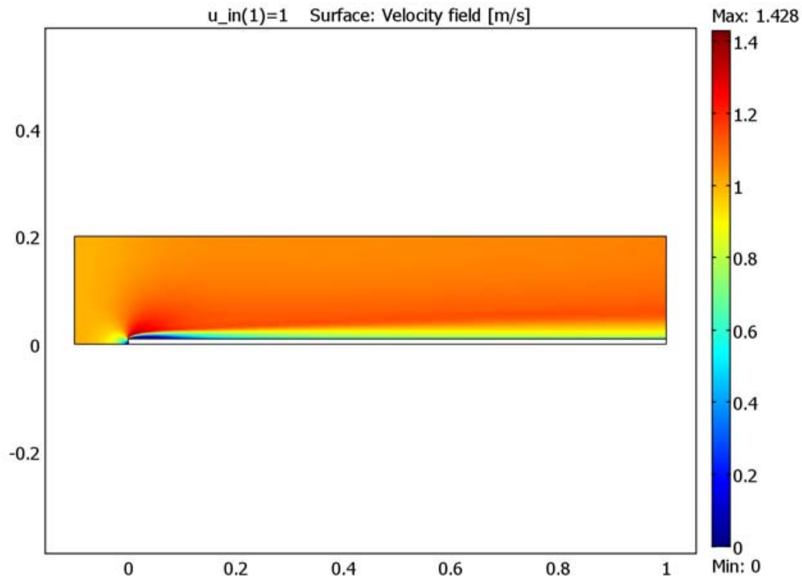


Figure 2-19: Velocity field at an inlet velocity of 1 m/s.

If you had treated the flow as being isothermal, the average would not have varied between the inlet and the outlet. However, for a nonisothermal flow the average velocity is inversely proportional to the average density, and it varies with changing average temperature. This means that the flow field for the fluid is different when taking density variations into account.

As the fluid heats up, its velocity increases slightly. Thus the boundary layer decreases and the local heat transfer coefficient should become larger. So if you neglect density variations when modeling forced convection cooling, the model slightly underestimates the cooling/heating power.

The accuracy in predicting the heat transfer coefficient in this example is dictated by the accuracy of the Reynolds analogy and the accuracy of the flow model. The situation this example models is very well studied, so you can readily verify the results in terms of heat transfer coefficient predictions. The following figure compares the local

h coefficient from the model with an empirical expression valid for the geometry and conditions studied (assuming turbulent flow).

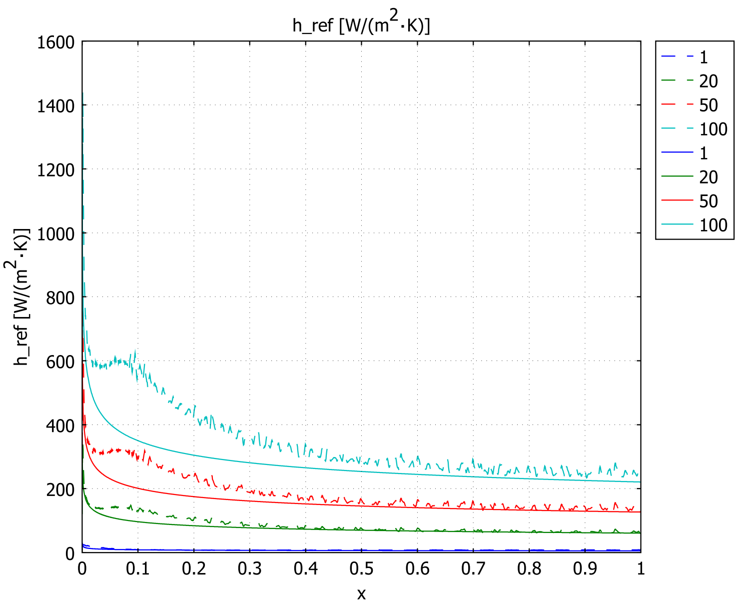


Figure 2-20: Local heat transfer coefficient as determined empirically (solid) and with the model (dashed) for various inlet velocities.

The model agrees well with empirical data for low to intermediate inlet velocities. The deviations at the leading edge of the plate are due to the correlation, which assumes that the boundary layer is fully developed for all x. However, at high inlet velocities the results do not match quite as well due to the model of the flow. The logarithmic wall function in COMSOL Multiphysics is valid under certain conditions that depend on the resolution, the velocity, and the viscosity. As mentioned previously on page 46, the wall function uses the dimensionless wall offset, δ_w^+ (defined in Equation 4-10 on page 59 if the *Heat Transfer Module User's Guide*). For the wall function to be an accurate approximation, δ_w^+ for the first internal node should be larger than 30 but less than some upper limit dependent on the Reynolds number (for more details see the section "Logarithmic Wall Function" on page 212 of the *Heat Transfer Module User's Guide*). Figure 2-21 depicts the parameter δ_w^+ against plate surface for various inlet velocities.

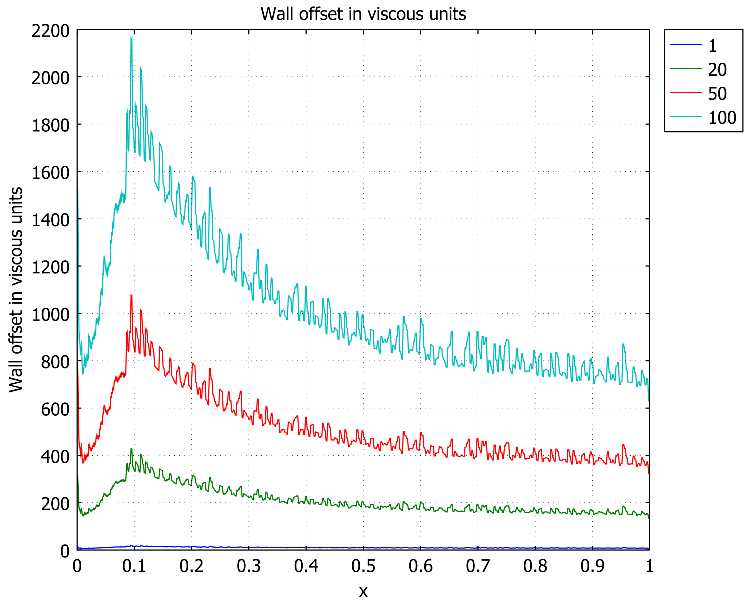


Figure 2-21: Dimensionless wall distance at the plate surface for various inlet velocities.

A maximum $\delta_w{}^+$ value of a few hundreds is always acceptable, whereas a value above 1000 is always questionable. Note that the value of $\delta_w{}^+$ exceeds 1000 when the inlet velocity is 50 m/s or higher. Hence, the mesh is a bit too coarse for this case. As a consequence, both the fluid velocity at the boundary and the heat transfer coefficient become less accurate. You can easily correct this situation by making the mesh finer at the boundary at the leading edge of the plate.

Results for the Simplified Model

Now examine the results of the much simpler model, which uses h coefficients from the built-in library. This simplified case does not model the flow field or the temperature distribution in the fluid. Therefore, it is “inexpensive” to solve in terms of memory requirements and calculation time—it solves in just a few seconds. Nevertheless, the results are rather accurate because the heat transfer coefficient is based on an empirical relationship.

Figure 2-22 compares the heat flux at the plate interface as calculated by this model to that of the previous, more complex, model.

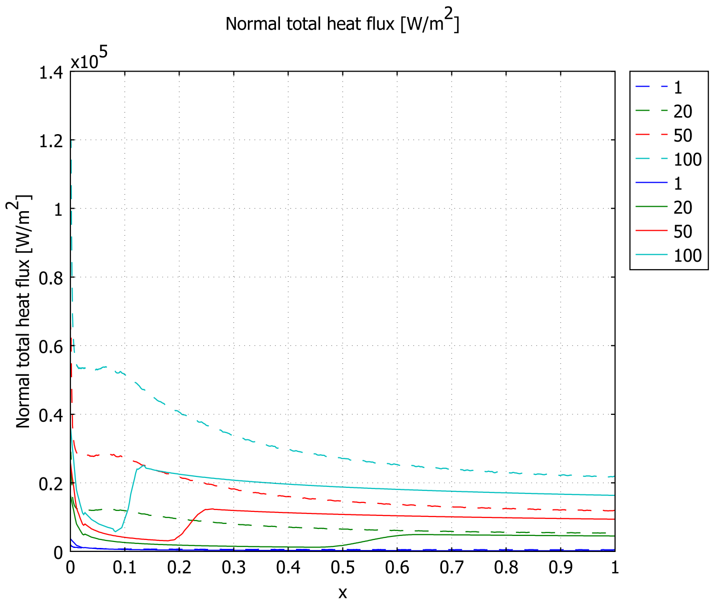


Figure 2-22: Comparison of normal total heat flux at the plate surface for the simplified (solid) and the flow/heat (dashed) models at various inlet air velocities.

The figure shows that the heat flux of the simplified model differs from the that of the flow/heat model. Because the simplified model uses well-established empirical relationships, you can consider its result more accurate. It shows the heat flux being significantly lower in the inlet region (at low values of x). This is a consequence of the initially laminar flow, which results in a much lower heat transfer coefficient. Then, above a certain x value, the flow turns turbulent and the heat transfer coefficient grows significantly. This appears in the plot as a sudden increase in the heat flux. On the other hand, the flow/heat model assumes that the flow is turbulent in the entire geometry, and therefore the heat flux is significantly larger.

To conclude, the flow/heat approach results in rather good predictions of the local boundary heat flux compared to reference values, but it assumes that the flow is turbulent at the inlet. That method is rather straightforward to model in COMSOL Multiphysics but requires a few minutes of computational time. On the other hand, the simplified approach is very powerful for situations where you are interested only in the solid's boundary heat flux. You can employ this approach, however, only if you can find a well-established correlation. For many geometries such correlations do not exist, and then the flow/heat approach is useful. The choice of method for modeling convective cooling or heating depend on your needs and the particular case.

References

1. A. Bejan, Heat Transfer, 1993, John Wiley.

2. B. Sundén, "Kompendium i Värmeöverföring," Department of Heat Transfer, LTH, Lund University, Sweden, p. 137, 2004 (in Swedish).

Modeling in COMSOL Multiphysics

The COMSOL Multiphysics implementation is straightforward using the Heat Transfer Module's Turbulent Non-Isothermal Flow multiphysics coupling, combining the General Heat Transfer and k-ε Turbulence Model application modes. In the following steps you begin by setting up and solving the model with the fluid included. In the next section you then simplify the model.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/forced_turbulent_convection`

Modeling Using the Graphical User Interface—The Flow/Heat Model

MODEL NAVIGATOR

1 Open the **Model Navigator**, and from the **Space dimension** list select **2D**.

2 In the list of application modes, select **Heat Transfer Module>Fluid-Thermal Interaction>Turbulent Non-Isothermal Flow, k-ε**.

3 Click **OK**.

GEOMETRY

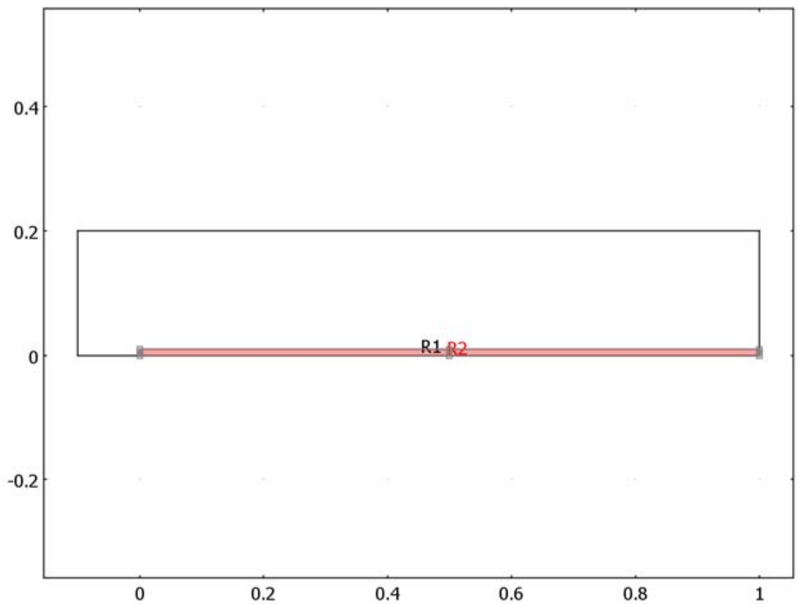
1 Using the **Rectangle** dialog box, create two rectangles with specifications according to the following table. You launch the **Rectangle** dialog box by shift-clicking the

Rectangle/Square button on the Draw toolbar or by choosing **Specify Objects>Rectangle** from the Draw menu.

OBJECT	WIDTH	HEIGHT	BASE	X	Y
R1	1.1	0.2	Corner	-0.1	0
R2	1	0.01	Corner	0	0

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

You should now see the following geometry:



CONSTANTS, EXPRESSIONS, AND VARIABLES

1 From the **Options** menu, open the **Constants** dialog box. Specify the following names, expressions, and descriptions (optional); when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_amb	293[K]	Surrounding air temperature
delta_T	100[K]	Plate-to-air temperature difference
T_av	T_amb+delta_T/2	Average temperature
p_ref	1.013e5[Pa]	Reference pressure
u_in	1[m/s]	Inlet velocity

2 Choose **Options>Expressions>Scalar Expressions**. Specify the following names, expressions, and descriptions (optional); when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
`L`	`x`	`Distance from leading edge`
`ReL_ref`	`u_in*L/mat1_nu0(T_av[1/K])[m^2/s]`	`Reference Reynolds number`
`Pr_ref`	`mat1_eta(T_av[1/K])[Pa*s]*mat1_Cp(T_av[1/K])[J/(kg*K)]/mat1_k(T_av[1/K])[W/(m*K)]`	`Reference Prandtl number`
`NuL_ref`	`0.037*ReL_ref^0.8*Pr_ref^0.33`	`Reference Nusselt number`
`h_ref`	`mat1_k(T_av[1/K])[W/(m*K)]*NuL_ref/L`	`Handbook h coefficient`

PHYSICS SETTINGS

Now it is time to set up the physics in the subdomain and the boundary conditions. In this model you load the material properties from the built-in materials library.

1 From the **Multiphysics** menu, select **3 k-ε Turbulence Model**.

2 From the **Physics** menu, select **Properties**.

3 Set **Realizability** to **On**, then click **OK**.

4 Choose **Physics>Subdomain Settings**. Select Subdomain 2 (the plate).

5 Select **Solid domain** from the **Group** list underneath the **Subdomain selection** list.

6 Select Subdomain 1 (the fluid) and click the **Load** button to open the **Materials/Coefficients Library** dialog box.

7 Select **Basic Material Properties>Air**, then click **OK**.

8 edit the predefined entry for the density. Click in the **Density** edit field, and replace `p` with `p_ref` and `T` with `Tf`; the entry should read `rho(p_ref[1/Pa],Tf[1/K])[kg/m^3]`.

9 Modify the expression in the **Dynamic viscosity** edit field by replacing `T` with `Tf`.

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

11 Choose **Physics>Boundary Settings**. Then apply the following boundary conditions:

SETTINGS	BOUNDARY 1	BOUNDARIES 2, 3	BOUNDARIES 4, 6	BOUNDARY 8
Type	Inlet	Symmetry boundary	Wall	Outlet
Condition	Velocity		Logarithmic wall function	Pressure
u_0	`u_in`			
L_T	`0.001`			
I_T	`0.01`			
δ_w			h	
p_0				0

The small values of L_T and I_T are appropriate for essentially non-turbulent free-stream flows.

12 Click **OK**.

Now set up the parameters for the heat transfer.

1. From the **Multiphysics** menu, select **1 General Heat Transfer (htgh)**.
2. From the **Physics** menu, select **Subdomain Settings**.
3. Select Subdomain 2. Then select **Solid domain** from the **Group** list.
4. Select Subdomain 1. From the **Library material** list, select **Air**.
5. For that material, edit the expressions for the **Thermal Conductivity**, the **Density**, and the **Heat capacity** by replacing `p` with `p_ref` and `T` with `Tf`.
6. Click the **Init** tab. In the **Tf(t_0)** edit field type `T_amb`, then click **OK**.
7. From the **Physics** menu, open the **Boundary Settings** dialog box.
8. Specify boundary conditions according to the following table. When done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARIES 2, 3	BOUNDARIES 4, 6	BOUNDARY 8
Object	Inlet	Top and inlet bottom boundary	Plate surface	Outlet
Group			wall	
Boundary condition	Temperature	Insulation/Symmetry		Convective flux
T_0	`T_amb`			

9. From the **Multiphysics** menu select, **2 General Heat Transfer (htgh2)**.
10. From the **Physics** menu, select **Subdomain Settings**.

11 Select Subdomain 2, then select **Solid domain** from the **Group** list. The default physical parameters correspond to copper and are correct. Click **OK**.

12 From the **Physics** menu, select **Boundary Settings**.

13 Specify the following boundary conditions; when finished, click **OK**.

SETTINGS	BOUNDARY 5	BOUNDARIES 4, 6
Object	Plate bottom (hot side)	Plate surface
Group		wall
Boundary condition	Temperature	
T_0	`T_amb+delta_T`	

MESH GENERATION

To solve the problem and get an accurate solution, the mesh must be fine at the solid/fluid interface, especially at the point of first contact. Generate such a mesh with the following steps:

1 Choose **Mesh>Free Mesh Parameters**. In the list of **Predefined mesh sizes** select **Coarse**.

2 Go to the **Boundary** page and select Boundaries 4 and 6. Set the **Maximum element size** to `3e-3` and the **Element growth rate** to `1.2`.

3 Go to the **Point** page and select Point 4. Set the **Maximum element size** to `1e-3`.

4 Click **Remesh** to generate the mesh. When the mesher has finished, click **OK**.

COMPUTING THE SOLUTION

Solve this model for a range of inlet velocities with the parametric solver. The solution procedure involves a first solution step that solves for the fluid velocity without any influence from temperature using only one inlet velocity. This is necessary to get a good initial value for the thermally coupled calculations. That solution then works in the parametric solver, which solves the problem as fully coupled for a set of inlet velocities.

1 Click the **Solver Parameters** button on the Main toolbar.

2 From the **Solver** list, select **Parametric segregated**.

3 Select the **Manual specification of segregated steps** check box.

4 Set the **Damping** for **Group 1** to `0.25`.

5 In the **Parameter name** edit field type `u_in`, and in the **Parameter values** edit field type `1 20 50 100`.

6 Click the **Parametric** tab. From the **Predictor** list, select **Constant**.

7 Select the **Manual tuning of parameter step size** check box. In the three edit fields (**Initial step size**, **Minimum step size**, and **Maximum step size**) type `2`, `20`, and `50`, respectively. These settings force the parameter solver to take larger steps than it would do by default, which in turn reduces the solution time.

8 Click **OK**, then click the **Solve** button on the Main toolbar. The software needs roughly 30 minutes to solve this setup on a 3-GHz PC.

POSTPROCESSING AND VISUALIZATION

Reproduce the plots in Figure 2-16–Figure 2-19 using the **Plot Parameters** dialog box.

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page, select **1** from the **Parameter value** list.

3 Click **Apply** to generate the plot in Figure 2-16.

Proceed to generate the plot in Figure 2-17 of the turbulent thermal conductivity, k_T with the following steps:

4 On the **General** page, select **100** from the **Parameter value** list.

5 Click the **Surface** tab. Type `kT_htgh` in the **Expression** edit field on the **Surface Data** page, then click **Apply** to generate the plot.

Next reproduce Figure 2-18 as follows:

6 While still on the **Surface** page, type `rho_htgh` in the **Expression** edit field on the **Surface Data** page.

7 On the **General** page, select **1** from the **Parameter value** list. Click **Apply**.

To generate Figure 2-19 execute the following instructions:

8 Click the **Surface** tab. From the **Predefined quantities** list on the **Surface Data** page, select **k-ε Turbulence Model (chns)>Velocity field**.

9 Click **OK**.

Use the **Domain Plot Parameters** dialog box to generate Figure 2-20–Figure 2-22:

1 From the **Postprocessing** menu, select **Domain Plot Parameters**.

2 On the **Line/Extrusion** page, select Boundary 6.

3 In the **y-axis data** area, type `abs(ntflux_htgh)/(Ts-Tf))` in the **Expression** edit field. From the **x-axis data** list, select **x**.

4 Click the **Line Settings** button. From the **Line style** list, select **Dashed line**. Select the **Legend** check box, then click **OK**.

5 Click **Apply** to generate the first lines of the plot.

6 On the **General** page, select the **Keep current plot** check box.

7 Return to the **Line/Extrusion** page. In the **Expression** edit field, type `h_ref`.

8 Click the **Line Settings** button. From the **Line style** list, select **Solid line**. Click **OK**.

9 Click **Apply** to finalize the plot in Figure 2-20.

Next, turn to the plot in Figure 2-21:

10 On the **General** page, clear the **Keep current plot** check box.

11 On the **Line/Extrusion** page, type `dwplus_chns` in the **Expression** edit field.

12 Click **Apply**.

Finally, you reproduce the plot in Figure 2-22 with the following steps:

13 From the **Predefined quantities** list select **General Heat Transfer (htgh2)>Normal total heat flux**.

14 Click the **Line Settings** button. From the **Line style** list select **Dashed line**, then click **OK**.

15 Click **OK** to close the **Domain Plot Parameters** dialog box and generate the plot.

If you want to overlay the results from the simplified model, keep the Figure 1 window open and then proceed as follows:

1 To open the other model, choose **File>Open Model Library**, browse to the location **Model Library>Heat Transfer Module>Electronics and Power Systems> forced_turbulent_convection_hcoeff**, and click **OK**.

2 From the **Postprocessing** menu, select **Domain Plot Parameters**.

3 On the **General** page, select **Figure 1** from the **Plot in** list and select the **Keep current plot** check box.

4 On the **Line/Extrusion** page, select Boundary 3.

5 Select **General Heat Transfer (htgh)>Normal total heat flux** from the **Predefined quantities** list.

6 From the **x-axis data** list select **x**, then click **OK**.

Modeling Using the Graphical User Interface—Simplified Model

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/forced_turbulent_convection_hcoeff`

You can build this model from scratch or modify the previous one. If you prefer to modify the previous model, simply alter the boundary settings of the second General Heat Transfer application mode and the solver settings so that you only solve for the htgh2 application mode.

To build the model from scratch, follow these steps:

MODEL NAVIGATOR

1 Open the **Model Navigator**, and from the **Space dimension** list select **2D**.

2 In the list of application modes select **Heat Transfer Module>General Heat Transfer>Steady-state analysis**. Click **OK**.

GEOMETRY

For this model it suffices to draw the plate because you model only the temperature in the solid plate.

Shift-Click the **Rectangle/Square** button on the Draw toolbar. Create a rectangle with the following specifications; when finished, click **OK**.

OBJECT	WIDTH	HEIGHT	BASE	X	Y
R1	1	0.01	Corner	0	0

CONSTANTS, EXPRESSIONS, AND VARIABLES

From the **Options** menu open the **Constants** dialog box. Specify the following names and expressions, then click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_amb	293[K]	Surrounding air temperature
delta_T	100[K]	Plate-to-air temperature difference

PHYSICS

The default material settings are those for copper. This means you do not have to modify the material properties. However, for the model to run smoothly, it is important to specify a suitable initial value in the domain.

1 Select the menu item **Physics>Subdomain Settings**.

2 Select Subdomain 1.

3 Click the **Init** tab. In the **T(t_0)** edit field type T_amb+delta_T, then click **OK**.

4 From the **Physics** menu open the **Boundary Settings** dialog box.

5 Select Boundary 2 (the bottom of the plate). In the **Boundary condition** list select **Temperature**, then in the $\mathbf{T_0}$ edit field type `T_amb+delta_T`.

6 Select Boundary 3 (the fluid interface that is cooled). In the **Boundary condition** list select **Heat flux**.

7 To load a heat transfer coefficient function from the Materials/Coefficients Library, click the **Load** button to open the **Materials/Coefficients Library** dialog box.

8 Select the coefficient function with the name **Forc. Plate, h_loc, s=position, U=velocity** from the **Heat Transfer Coefficients>Air, Ext. Forced Convection** folder; then click **OK**.

9 When back in the **Boundary Conditions** dialog box, edit the last two input variables to the `h_loc` function so that the value in the **h** edit field becomes `h_loc(T[1/K],Tinf_htgh[1/K],x[1/m],u_in)[W/(m^2*K)]`. The *x* coordinate and the inlet velocity replace the default input variables for length and velocity.

10 In the $\mathbf{T_{inf}}$ edit field and type `T_amb`, then click **OK**.

SOLUTION AND POSTPROCESSING

1 Open the menu item **Solve>Solver Parameters**. From the **Solver** list select **Parametric**.

2 Go to the **Parameter name** edit field and type `u_in`. In the **Parameter values** edit field type `1 20 50 100`.

3 Click the **Parametric** tab. In the **Predictor** list select **Constant**.

4 Select **Manual tuning of parameter step size** check box. In all three edit fields (**Initial step size**, **Minimum step size**, and **Maximum step size**) type `50`. This setting forces the parameter solver to take steps as large as possible, which reduces the solution time.

5 Click **OK**, then click the **Solve** button on the Main toolbar. The model solves in a few seconds.

6 To generate the part of Figure 2-22 related to these results, select **Postprocessing>Domain Plot Parameters**. Go to the **Line/Extrusion** page.

7 Select Boundary 3. Select **Normal total heat flux (htgh)** from the list of **Predefined quantities**.

8 From the **x-axis data** list select **x**, then click **OK**.

Surface-Mount Resistor

Introduction

The drive for miniaturizing electronic devices has resulted in today's extensive use of surface-mount electronic components. An important aspect in electronics design and the choice of materials is a product's durability and lifetime. For surface-mount resistors and other components producing heat it is a well-known problem that temperature cycling can lead to cracks propagating through the solder joints, resulting in premature failure (Ref. 1). For electronics in general there is a strong interest in changing the soldering material from lead- or tin-based solder alloys to other mixtures.

The following multiphysics example models the heat transport and structural stresses and deformations resulting from the temperature distribution using the General Heat Transfer application mode and the Solid, Stress-Strain application mode.

Model Definition

Figure 2-23 shows a photograph of a surface-mount resistor together with a diagram of it on a printed circuit board (PCB).

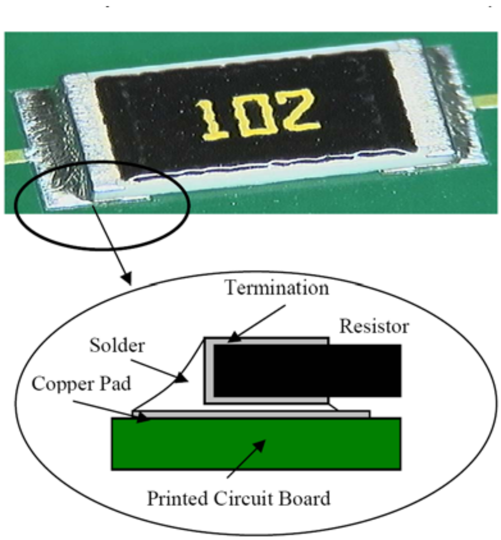


Figure 2-23: A photo and diagram of a typical surface-mounted resistor soldered to a PCB.

Table 2-2 shows the dimensions of the resistor and other key components in the model including the PCB.

TABLE 2-2: COMPONENT DIMENSIONS

COMPONENT	LENGTH	WIDTH	HEIGHT
Resistor (Alumina)	6 mm	3 mm	0.5 mm
PCB (FR4)	16 mm	8 mm	1.6 mm
Cu pad	2 mm	3 mm	35 µm
Ag termination	0.5 mm	3 mm	25 µm
Stand-off (gap to PCB)	-	-	105 µm

The simulation uses a symmetry cut along the length of the resistor so that it needs to include only half of the component (Figure 2-24).

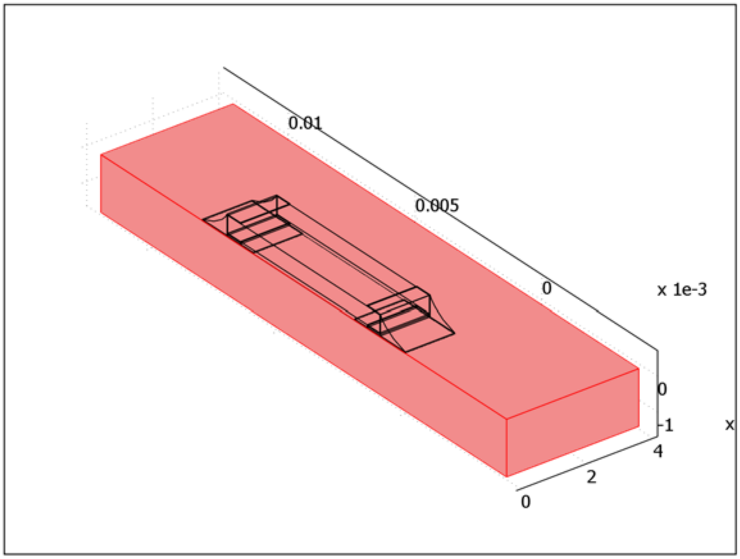


Figure 2-24: The simulation models only half the resistor.

In operation, the resistor dissipates 0.2 W of power as heat. Conduction to the PCB and convection to the surrounding air provide cooling. In this model, the heat transfer occurs through conduction in the subdomains. The model simplifies the surface cooling and describes it using a heat transfer coefficient, h, in this case set to 5 W/(m^2·K); the surrounding air temperature, T_{inf}, is at 300 K. The resulting heat-transfer equation and boundary condition (included in the model using the General Heat Transfer application mode) are

$$\nabla \cdot (-k\nabla T) = Q$$

$$-\mathbf{n} \cdot (-k\nabla T) = h(T_{inf} - T)$$

where k is the thermal conductivity, and Q is the heating power per unit volume of the resistor (set to 16.7 MW/m^3 corresponding to 0.2 W in total).

The model handles thermal expansion using a static structural analysis using the Solid, Stress-Strain application mode (a description of the corresponding equations is available in the *Structural Mechanics Module User's Guide*). The thermal and mechanical material properties in this model are:

TABLE 2-3: MATERIAL PROPERTIES

MATERIAL	E (GPa)	n	α (ppm)	k (W/(m·K))	ρ (kg/m^3)	C_p (J/(kg·K))
Ag	83	0.37	18.9	420	10500	230
Alumina	300	0.222	8.0	27	3900	900
Cu	110	0.35	17	400	8700	385
Fr4	22	0.28	18	0.3	1900	1369
60Sn-40Pb	10	0.4	21	50	9000	150

The model treats properties of air as temperature dependent according to the following equations (Ref. 3):

$$\rho = \frac{p_0 M_\mathrm{w}}{RT}$$

with $p_0 = 1$ atm, $M_\mathrm{w} = 0.0288$ kg/mol, and $R = 8.314$ J/(mol·K). Further,

$$C_p = 1100 \ \mathrm{J/(kg{\cdot}K)}$$

$$k = 10^{-3.723 + 0.865\log(T)} \ \mathrm{W/(m{\cdot}K)}$$

The stresses are zero at 293 K. The boundary condition for the Solid, Stress-Strain application mode is that both ends, in the length direction of the circuit board, are fixed with respect to x, y, and z.

Note: This model requires the Heat Transfer Module and the Structural Mechanics Module.

Results and Discussion

The isosurfaces in Figure 2-25 show the temperature distribution at steady state. The highest temperature is approximately 424 K, appearing in the center of the resistor. The circuit board also heats up significantly.

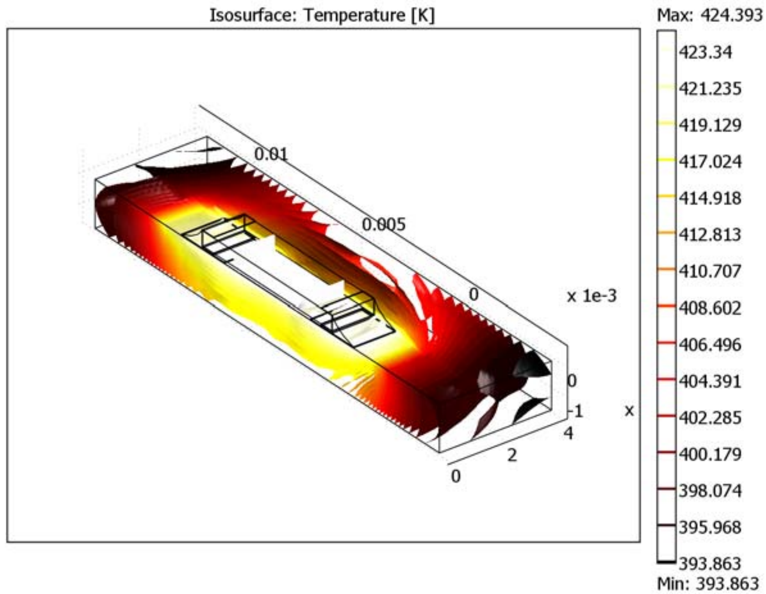


Figure 2-25: Temperature distribution in the resistor and the circuit board at steady state.

Thermal stresses appear as a result of the temperature increase; they arise from the materials' different expansion coefficients. Figure 2-26 plots the effective stress (von Mises) together with the resulting deformation of the assembly.

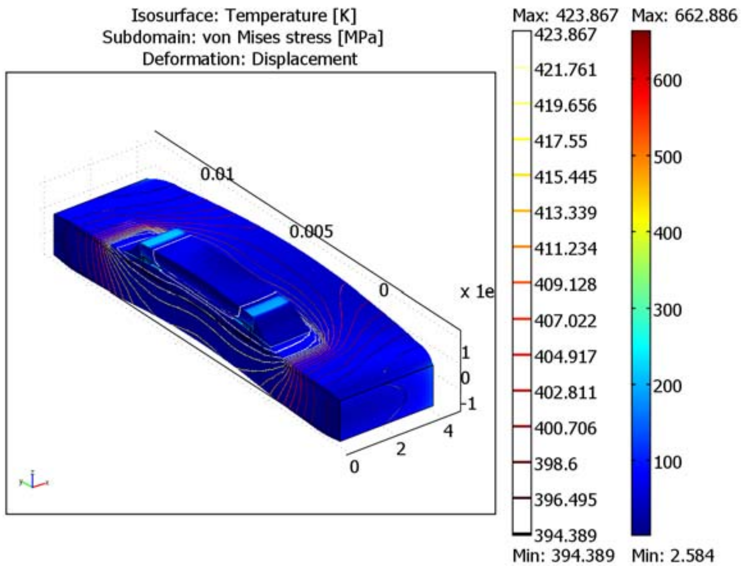


Figure 2-26: The thermally induced distribution of von Mises effective stress together with the deformation (magnified) and the isotherms.

The highest stresses seem to occur in the termination material. It is interesting to compare these effective stresses to the yield stress and thereby investigate whether or not the material is irreversibly deformed. In that case the solder is the weak point. The following graph plots the stress in the solder points alone.

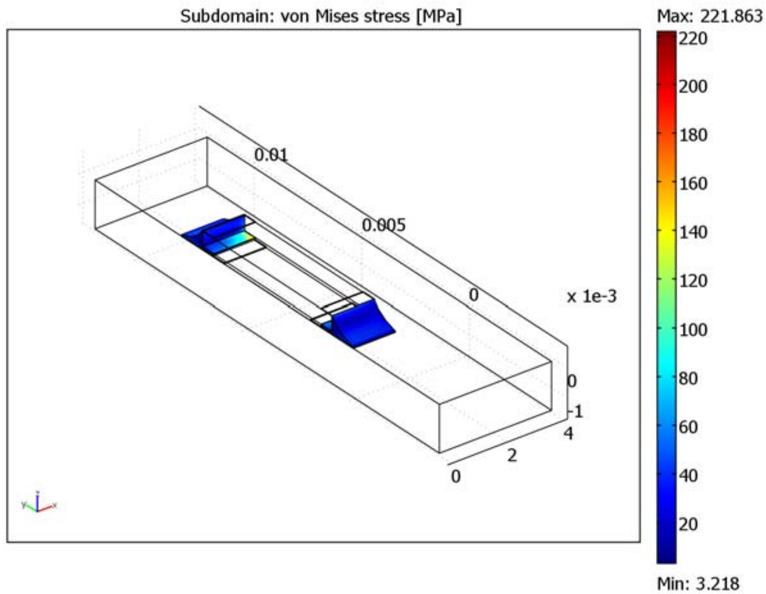


Figure 2-27: Close-up of the von Mises effective stresses in the solder joint.

The yield stress for solder is approximately 220 MPa. The highest effective stress seems to fall in the range near 220 MPa. This means that the assembly functions without failure for the tested power loads. However, if the heating power increases slightly, permanent deformation and possibly failure occur.

References

1. H.Lu, C.Bailey, M.Dusek, C.Hunt, and J.Nottay, "Modeling the Fatigue Life of Solder Joints of Surface Mount Resistors," EMAP 2000.

2. Courtesy of Dr. H. Lu, Centre for Numerical Modelling and Process Analysis, University of Greenwich, U.K.

3. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, Pergamon Press, 1990, appendix.

Modeling in COMSOL Multiphysics

Use the predefined multiphysics coupling Thermal-Structure Interaction to set up the model. The General Heat Transfer application mode and the Solid, Stress-Strain application mode are then automatically added to the model, with the temperature variable from the heat equation predefined as the strain temperature in the equation for the structural deformations.

Take advantage of the assembly functionality and interactive meshing tools to create a swept mesh for the geometry; in this way you can keep the memory requirements for solving the problem at a minimum.

The default solver settings use the segregated stationary solver to solve the heat equation first, because it does not depend on the structural analysis. The GMRES iterative solver with the Geometric multigrid preconditioner is the default for the heat equation. This iterative solver requires less memory for solving the equation than a direct solver. Select the same solver combination also for the structural mechanics equation to further reduce the memory needed. In order that it work well with the mesh of the model, you can tune the Geometric multigrid preconditioner; read more in the sections “The GMRES Iterative Solver” on page 548 of the *COMSOL Multiphysics Reference Guide* and “The Preconditioned Linear System” on page 435 of the *COMSOL Multiphysics User’s Guide*.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/surface_resistor`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**.

2 On the **New** page, select **3D** from the **Space dimension** list.

3 Select **Structural Mechanics Module>Thermal-Structural Interaction>Solid, Stress-Strain with Thermal Expansion>Static analysis**.

4 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2 Define the following constants (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_air	293[K]	Air temperature
h_air	5[W/(m^2*K)]	Heat transfer coefficient
q_source	0.2[W]/(0.5*3*8[mm^3])	Heating power per unit volume
p0	1[atm]	Air pressure

GEOMETRY MODELING

1 From the **Draw** menu, select **Work Plane Settings**.

2 On the **Quick** page, click the **y-z** option button in the **Plane** area.

3 Click **OK**.

COMSOL Multiphysics adds a new 2D geometry to the model. Use this geometry to draw the cross section of the resistor geometry.

4 Create four rectangles. Shift-click the **Rectangle/Square** button on the Draw toolbar for each one and enter the data from this table:

RECTANGLE	WIDTH	HEIGHT	X	Y
R1	0.002	35e-6	0	0
R2	5.25e-4	4.5e-5	9.75e-4	3.5e-5
R3	5.25e-4	5.5e-4	9.75e-4	8e-5
R4	6e-3	5e-4	1e-3	1.05e-4

5 Click the **Zoom Extents** button on the Main toolbar.

6 Copy rectangle R4 by selecting it and then pressing Ctrl+C.

7 Click the **Create Composite Object** button on the Draw toolbar. (Alternatively, select **Create Composite Object** from the **Draw** menu.)

8 In the **Set formula** edit field type `R3-R4`, then click **OK**.

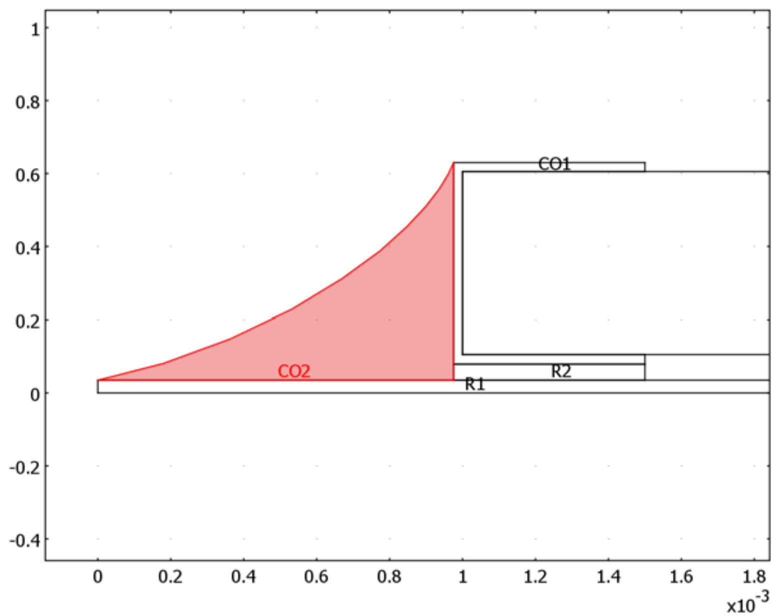
9 Paste a copy of R4 with zero displacement. To do so, press Ctrl+V, then click **OK** in the **Paste** dialog box.

10 Click the **2nd Degree Bézier Curve** button on the Draw toolbar.

11 Draw a curve between the upper corner of the termination and the left corner of the copper plate as in the figure below. You may want to zoom in the area before you start drawing the curve. Draw the line by clicking the coordinates

$(9.75 \cdot 10^{-4}, 6.3 \cdot 10^{-4})$, $(8 \cdot 10^{-4}, 2 \cdot 10^{-4})$, and $(0, 3.5 \cdot 10^{-5})$. The coordinates that the mouse is pointing to appears in the lower left corner of the user interface.

12 After clicking the third coordinate pair click the **Line** button on the Draw toolbar. This allows you to continue the drawing with lines along the copper plate boundary and the termination boundary**.** Click on the coordinates $(9.75 \cdot 10^{-4}, 3.5 \cdot 10^{-5})$ and $(9.75 \cdot 10^{-4}, 6.3 \cdot 10^{-4})$. Then complete the drawing by right-clicking using the mouse. The drawing should now look like in this figure:



13 Copy the objects R1, R2, CO1, and CO2 by selecting them and pressing Ctrl+C.

14 Press Ctrl+V to open the **Paste** dialog box. Go to the **Displacement** area and in the **x** edit field type `0.006`. Click **OK**.

15 From the **Draw** menu, select **Modify>Scale**. Find the **Scale factor** area, then in the **x** edit field type `-1`. Go to the **Scale base point** area and in the **x** edit field type `0.007`. Click **OK**.

16 Click the **Zoom Extents** button on the Main toolbar.

17 Click the **Line** button on the Draw toolbar and draw a line between the coordinates $(0.002, 0)$ and $(0.006, 0)$.

18 To finalize the geometry select all the objects by pressing Ctrl+A and click the **Coerce to Solid** button on the Draw toolbar.

19 Shift-click the **Rectangle/Square** button. Specify settings according to the following table. When done, click **OK**.

WIDTH	HEIGHT	BASE	X	Y
`16e-3`	`1.6e-3`	Center	`4e-3`	`-8.0e-4`

20 From the **Draw** menu, open the **Extrude** dialog box.

21 From the **Objects to extrude** list, select **CO5**. In the **Distance** edit field type `1.5e-3`, then click **OK**.

22 Return to the 2D geometry by clicking the **Geom2** tab.

23 From the **Draw** menu, open the **Extrude** dialog box.

24 Select **R1** from the **Objects to extrude** list and type `4e-3` in the **Distance** edit field. Click **OK**.

MESH GENERATION

1 From the **Draw** menu, select **Create Pairs**. Select both EXT1 and EXT2. Clear the **Create imprints** check box so that the meshes of the two extruded objects do not have to match at their shared boundary. This makes the mesh generation easier and reduces the number of elements.

2 Click **OK**.

3 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box and click the **Boundary** tab.

4 Select Boundaries 7, 10, 13, 16, 33, 37, 40, and 48, then click the **Mesh Selected** button.

5 Now select Boundaries 20 and 24 and click the **Mesh Selected** button again.

6 Click **OK** to close the **Free Mesh Parameters** dialog box.

7 Open the **Mapped Mesh Parameters** dialog box from the **Mesh** menu, then click the **Edge** tab.

8 Select **1** from the **Edge selection** list. Select the **Constrained edge element distribution** check box and set the **Number of edge elements** to `5`.

9 Repeat Step 8 for Edge 2, and set the number of edge elements to 30.

10 Click the **Boundary** tab and select **1** from the **Boundary selection** list.

11 Click **Mesh Selected**, then click **OK**.

12 From the **Mesh** menu, open the **Swept Mesh Parameters** dialog box.

13 Select Subdomain 1, then press Ctrl+A to select all subdomains.

14 Select the **Manual specification of element layers** check box and set the **Number of element layers** to 10. Click **OK**.

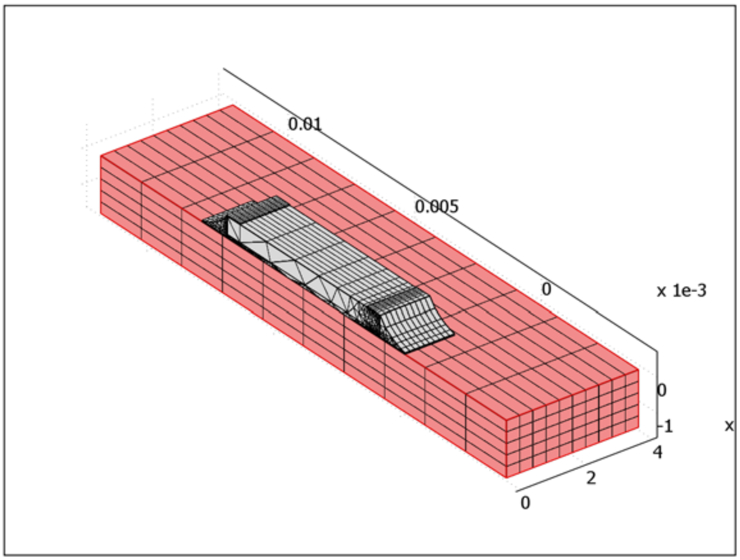
15 Click the **Subdomain Mode** button on the Main toolbar.

16 Select only the PCB domain, that is, the largest subdomain.

17 Click **Mesh>Interactive Meshing>Mesh Selected (Swept)**.

18 Click **Mesh>Interactive Meshing>Mesh Remaining (Swept)**.

The meshed geometry in the drawing area should now look like that in the following figure:



PHYSICS SETTINGS

Subdomain Settings

1 From the **Multiphysics** menu, select **Geom2: Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select all subdomains, then click the **Load** tab (not the **Load** button). In the **Tempref** edit field, type `T_air`.

4 Click the **Material** tab.

5 Select Subdomain 1. Click the **Load** button to open the **Materials/Coefficients Library** dialog box. Select **Basic Material Properties>FR4 (Circuit Board)**. Click **OK**.

6 Repeat the previous step for the other subdomains with materials according to the following table (the material data for silver, Ag, is available in the Metals section of the MEMS Material Properties library):

PROPERTY	SUBDOMAINS 2, 8	SUBDOMAINS 3, 4, 9, 11	SUBDOMAINS 5, 10	SUBDOMAIN 6
Material	Copper	Solder, 60Sn-40Pb	Ag	Alumina

7 Select Subdomain 7, then clear the **Active in this domain** check box.

8 Click **OK** to close the dialog box.

9 Change the active application mode. From the **Multiphysics** menu select **Geom2: General Heat Transfer (htgh)**.

10 From the **Physics** menu, select **Subdomain Settings**. Go to the **General** page, select Subdomain 1, and then select **FR4 (Circuit Board)** from the **Library material** list.

11 Repeat for the other subdomains according to:

PROPERTY	SUBDOMAINS 2, 8	SUBDOMAINS 3, 4, 9, 11	SUBDOMAINS 5, 10	SUBDOMAIN 6	SUBDOMAIN 7
Material	Copper	Solder, 60Sn-40Pb	Ag	Alumina	Air
Q	0	0	0	`q_source`	0

For Subdomain 7, change the name of the pressure variable in the expression for the density from `p` to `p0`, the air pressure, so that the expression in the ρ edit field becomes `rho(p0[1/Pa],T[1/K])[kg/m^3]`.

12 Go to the **Init** page. Select all subdomains and in the **Temperature** edit field, type `T_air`. Click **OK**.

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select the exterior boundaries in contact with air, that is, Boundaries 3, 4, 8, 12, 19, 29, 30, 44, 46, and 52–63. From the **Boundary condition** list, select **Heat flux**.

3 In the **Heat transfer coefficient** edit field type `h_air`, and in the **External temperature** edit field type `T_air`. Click **OK**.

4 In the **Multiphysics** menu, change the active application mode to **Geom2: Solid, Stress-Strain (smsld)**.

5 From the **Physics** menu, open the **Boundary Settings** dialog box.

6 Select Boundaries 1, 7, 10, 13, 16, 20, 33, 37, 40, and 48. From the **Constraint condition** list, select **Symmetry plane**.

7 Select Boundaries 2 and 5. From the **Constraint condition** list, select **Fixed**. Click **OK**.

COMPUTING THE SOLUTION

Change the default solver settings to reduce the memory requirements for solving the model.

1 Click the **Solver Parameters** button on the Main toolbar.
2 Locate the **Segregated groups** area on the **General** page and click the **Settings** button for Group 1.
3 In the **Linear System Solver Settings** dialog box that opens, choose **Right** from the **Preconditioning** list; click **OK**.
4 Click the **Settings** button for Group 2.
5 In the dialog box that opens, select **GMRES** from the **Linear system solver** list.
6 From the **Preconditioning** list, select **Right**.
7 In the **Linear system solver** tree on the left, select **Preconditioner**.
8 From the **Preconditioner** list, select **Geometric multigrid**.
9 Click **OK** to close the **Linear System Solver Settings** dialog box.
10 Click **OK** to close the **Solver Parameters** dialog box.
11 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To reproduce the temperature plot in Figure 2-25, follow these instructions:

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.
2 Click the **General** tab.
3 In the **Plot type** area, clear the **Slice** check box and select the **Isosurface** check box.
4 Click the **Isosurface** tab.
5 From the **Predefined quantities** list, select **General Heat Transfer (htgh)>Temperature**.
6 In the **Isosurface levels** area, click the **Levels** button, then type 30 in the **Number of levels** edit field.
7 In the **Isosurface color** area, set the **Colormap** to **hot**.
8 Click **Apply**.
9 Click the **Scene Light** button on the Camera toolbar to finish off the plot.

Reproduce Figure 2-26 with these steps:

1 While still on the **Isosurface** page in the **Plot Parameters** dialog box, change the **Number of levels** to 15.
2 In the **Coloring and fill** area, set the **Fill style** to **Wireframe**.
3 Click the **Subdomain** tab and enable this plot type by selecting the **Subdomain plot** check box at the top of the dialog box. From the **Predefined quantities** list, select **Solid, Stress-Strain (smsld)>von Mises stress**. From the **Unit** list, select **MPa**.
4 Click the **Deform** tab and select the **Deformed shape plot** check box.
5 In the **Domain types to deform** area clear the **Boundary** and **Edge** check boxes.
6 In the **Deformation data** area click the **Subdomain Data**.
7 From the **Predefined quantities** list, select **Solid, Stress-Strain (smsld)>Displacement**.
8 Click **OK**.

Finally, to reproduce Figure 2-27, do the following:

1 From the **Options** menu, select **Suppress>Suppress Subdomains**.
2 Select Subdomains 1, 2, 5, 6, 7, 8, and 10, then click **OK**.
3 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.
4 On the **General** page, clear the **Isosurface** and **Deformed Shape** check boxes in the **Plot type** area.
5 Click **OK** to generate the plot.

Free Convection in a Light Bulb

Introduction

This model simulates the nonisothermal flow of argon gas inside a light bulb. The purpose of the model is to show the coupling between energy transport—through conduction, radiation, and convection—and momentum transport induced by density variations in the argon gas.

Model Definition

A light bulb contains a tungsten filament that is resistively heated when a current is conducted through it. At temperatures around 2000 K the filament starts to emit visible light. To prevent the tungsten wire from burning up, the bulb is filled with a gas, usually argon. The heat generated in the filament is transported to the surroundings through radiation, convection, and conduction. As the gas heats up, density and pressure changes induce a flow inside the bulb.

Figure 2-28 shows a cross section of the axially symmetric model geometry.

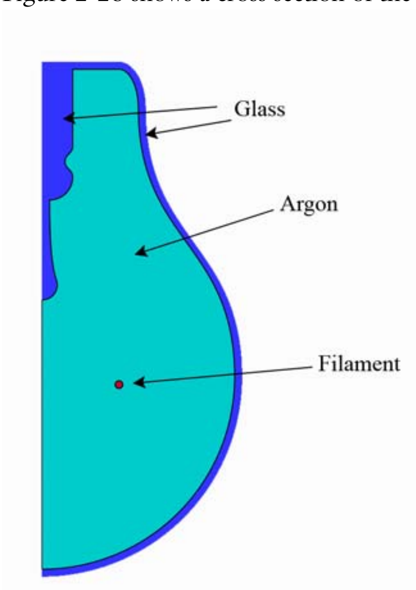


Figure 2-28: The model geometry.

The filament is approximated with a solid torus, an approximation that implies neglecting any internal effects inside the filament wire.

The equations governing the nonisothermal flow in the argon gas are

$$\rho\frac{\partial u}{\partial t} + \rho(\mathbf{u}\cdot\nabla\mathbf{u}) = -\nabla p + \nabla\cdot\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2\eta}{3}(\nabla\cdot\mathbf{u})\mathbf{I} + \rho\mathbf{g}$$

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$$

where ρ denotes the density (kg/m^3), $\mathbf{u}$ the velocity (m/s), η the viscosity (Pa·s), p the pressure (Pa), and $\mathbf{g}$ the gravity vector (m/s^2). The density is given by the ideal gas law

$$\rho = \frac{Mp}{RT}$$

where M denotes the molar weight (kg/mol), R the universal gas constant (J/(mol·K)) and T the temperature (K).

The convective and conductive heat transfer inside the bulb is described by the equation

$$\rho C_p\frac{\partial T}{\partial t} + \nabla\cdot(-k\nabla T) = -\rho C_p\mathbf{u}\cdot\nabla T + Q$$

where C_p denotes the heat capacity (J/(kg·K)), k is the thermal conductivity (W/(m·K)), and Q refers to the power density (W/m^3) in the filament that serves as a heat source. Assuming a total light bulb power of 60 W and a uniform power density leads to

$$Q = \frac{60\ \mathrm{W}}{\pi r^2 2\pi R_0}$$

where r is the filament's radius and $2\pi R_0$ gives its perimeter around the bulb.

BOUNDARY CONDITIONS

At the bulb's inner surfaces, radiation is described by surface-to-surface radiation. This means the mutual irradiation from the surfaces that can be seen from a particular surface, and radiation to the surroundings are accounted for. At the outer surfaces of the bulb, radiation is described by surface-to-ambient radiation, which means that there is no reflected radiation from the surroundings (blackbody radiation).

The top part of the bulb where the bulb is mounted on the cap is assumed to be insulated:

$$-\mathbf{n} \cdot (-k\nabla T) = 0$$

Results

The heating inside the bulb has a long and a short time scale from $t = 0$, when the light is turned on. The shorter scale captures the heating of the filament and the gas close to it. The following series of pictures shows the temperature distribution inside the bulb at $t = 2$, 6, and 10 s.

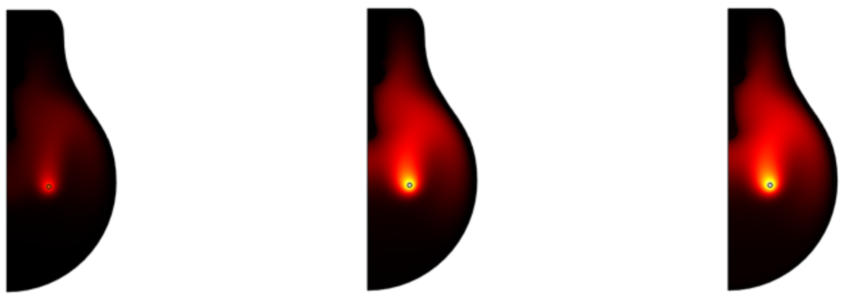


Figure 2-29: Temperature distribution at t = 2, 6, and 10 s. The temperature ranges between 298 (black) and 2010 K (yellow).

When the temperature changes, the density of the gas changes, inducing a gas flow inside the bulb. The following series of pictures shows the velocity field inside the bulb after 2, 6, and 10 s.

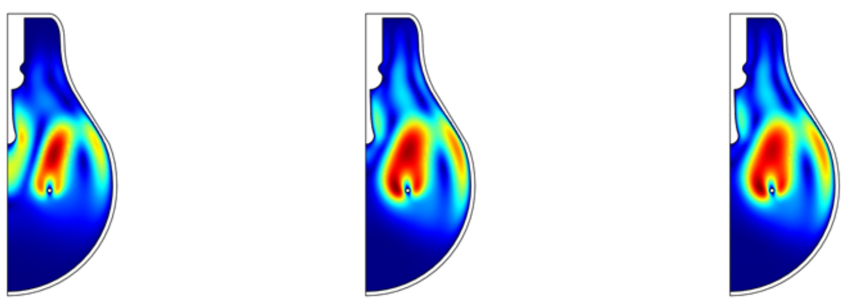


Figure 2-30: Velocity field after 2, 6, and 10 s. The velocity ranges between 0 and 0.24 m/s.

On the longer time scale, the glass on the bulb's outer side heats up. The following plot shows the temperature distribution in the bulb after 5 minutes.

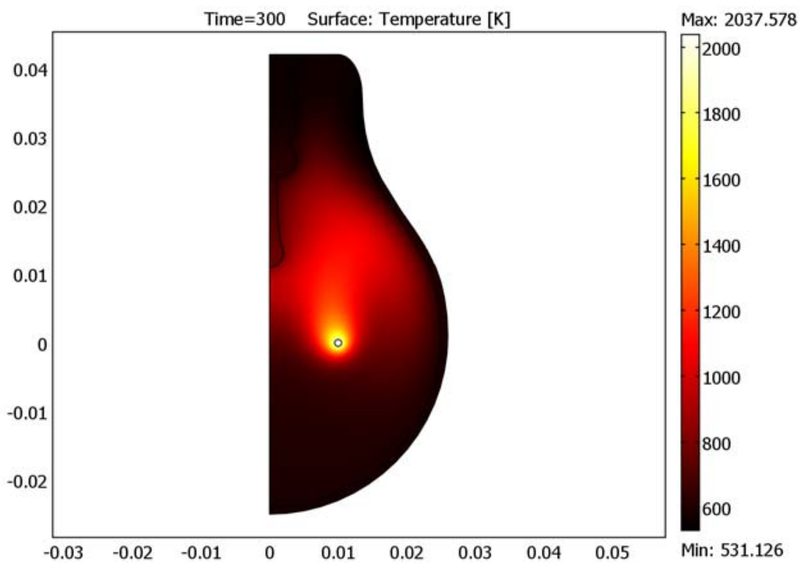


Figure 2-31: Temperature distribution after 5 minutes.

Figure 2-32 shows the temperature distribution at a point on the boundary of the bulb at the same vertical level as the filament. This plot shows the slow heating of the bulb. After 5 minutes, the bulb has reached a steady-state temperature of 580 K.

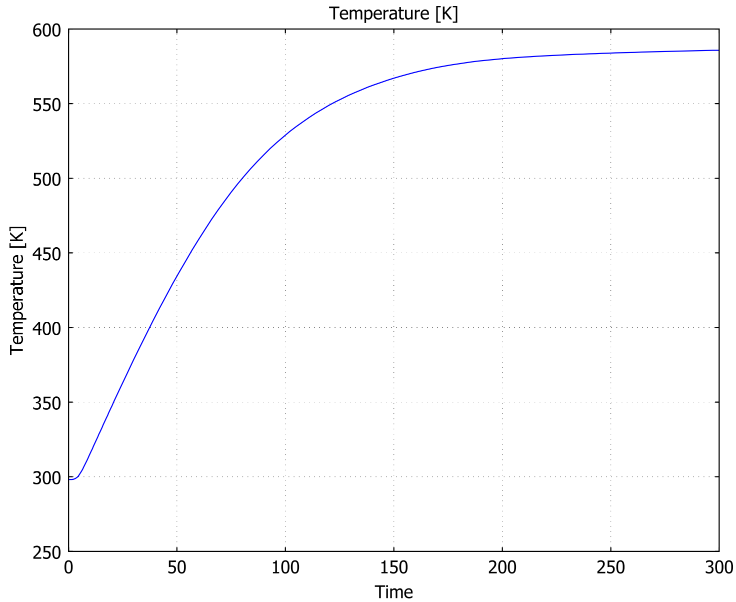


Figure 2-32: Temperature distribution at a point on the boundary of the bulb at the same vertical level as the filament.

Heat is transported from the boundary of the bulb through both convective heat flux and radiation. The net radiative heat flux leaving the bulb at $t = 300$ s is plotted in Figure 2-33, as function of the z-coordinate. The top boundaries of the bulb where the bulb is mounted on the cap are excluded from this plot. The distinct bump in the curve occurs at the same vertical level as the filament.

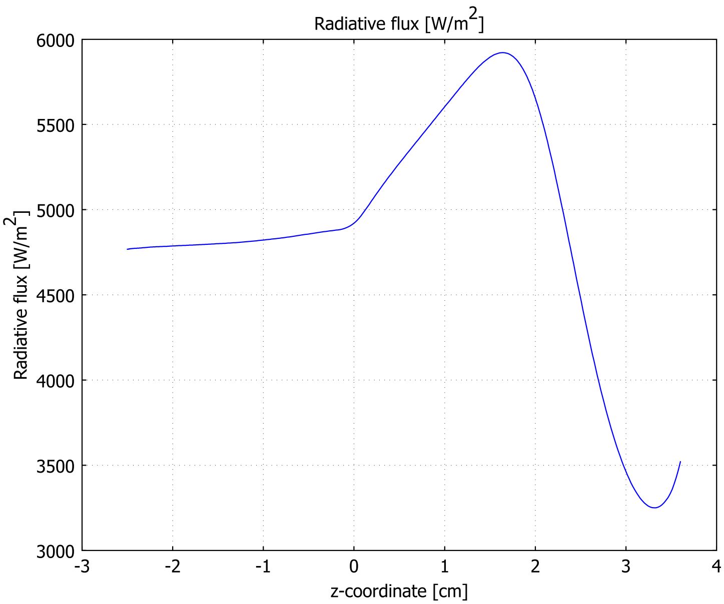


Figure 2-33: The net radiative heat flux leaving the bulb.

Modeling in COMSOL Multiphysics

To set up the model, use the Heat Transfer Module's Non-Isothermal Flow predefined multiphysics coupling. The model uses material from the Material Library to accurately account for temperature-dependent properties over a wide range. The model setup is straightforward and also shows how to create your own material to treat argon as an ideal gas. When working with surface-to-surface radiation in COMSOL Multiphysics, you have to specify which subdomains are transparent and which are opaque. Furthermore, to allow for surface-to-surface radiation, an opaque subdomain must be adjacent to a transparent subdomain. To achieve this, you can set the argon gas to be transparent and the glass and filament to be opaque. The assumption that the glass on the bulb is opaque, might seem odd, but is in fact valid because glass is almost opaque to heat radiation, but transparent to radiation in the visible spectrum.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/light_bulb`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. In the **Model Navigator**, select **Axial symmetry (2D)** from the **Space dimension** list.
2. From the **Application Modes** list, select **Heat Transfer Module>Fluid-Thermal Interaction>Non-Isothermal Flow>Transient analysis**.
3. Click **OK**.

GEOMETRY IMPORT

The model geometry is available as a CAD file. Import it to the COMSOL Multiphysics user interface by following these steps:

1. Choose **File>Import>CAD Data From File**.
2. Locate the COMSOL Multiphysics installation directory on your system, browse to the folder `models/Heat_Transfer_Module/Tutorial_Models`, and select the file `light_bulb.mphbin`.
3. Click **Import**.

OPTIONS AND SETTINGS

1. From the **Options** menu, select **Constants**.
2. Enter constants according to the following table (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
rho_glass	2595[kg/m^3]	Density, glass
k_glass	1.09[W/(m*K)]	Thermal conductivity, glass
cp_glass	750[J/(kg*K)]	Heat capacity, glass
h0	5[W/(m^2*K)]	Heat transfer coefficient
eps_glass	0.8	Surface emissivity, glass
vol	pi*(0.5[mm])^2*2*pi*10[mm]	Filament volume
Q	60[W]/vol	Heat source in filament

NAME	EXPRESSION	DESCRIPTION
g	9.81[m/s^2]	Gravitational constant
R	8.314[J/(mol*K)]	Universal gas constant
p0	50[kPa]	Initial pressure
k_tungsten	179[W/(m*K)]	Thermal conductivity, tungsten
Mw_a	39.94[g/mol]	Molar mass, argon (ideal gas)

3 From the **Options** menu, open the **Scalar Expressions** dialog box.

4 Specify the density according to the following table; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
rho_a	p*Mw_a/(R*T)	Density of argon, ideal gas

PHYSICS SETTINGS—GENERAL HEAT TRANSFER

1 Open the **Subdomain Settings** dialog box.

2 Select all Subdomains and click the **Init** tab.

3 In the **Temperature** edit field, type `25[degC]`.

4 Click the **Convection** tab.

5 Select Subdomains 1 and 3.

6 Clear the **Enable convective heat transfer** check box.

7 Select Subdomain 2.

8 In the **Absolute pressure** edit field, type `p`.

9 Click the **General** tab.

10 Select Subdomain 1.

11 Enter parameters according to the following table:

QUANTITY	VALUE/EXPRESSION	DESCRIPTION
k	k_glass	Thermal conductivity
ρ	rho_glass	Density
C_p	cp_glass	Heat capacity at constant pressure

12 Select Subdomain 3.

13 Click the **Load** button to open the **Materials/Coefficients Library**.

14 In the **Materials** area, select **Material Library>Elements>Tungsten**.

15 Click **OK**.

16 In the **Thermal conductivity** edit field, type `k_tungsten`.

17 In the **Heat source** edit field, type `Q`.

18 Select Subdomain 2.

19 Click the **Load** button to open the **Materials/Coefficients Library**.

20 Select **Material Library>Elements>Argon**.

21 From the **Phase/Condition** list, select **gas**.

22 Click **OK**.

23 From the **Opacity** list, select **Transparent**.

24 Click **OK**.

When modeling surface-to-surface radiation, it is important that a solid Subdomain is adjacent to a transparent Subdomain. In this case, the argon gas is modeled as a transparent domain.

1 Open the **Boundary Settings** dialog box.

2 Select Boundaries 1–6.

3 From the **Boundary condition** list, select **Axial symmetry**.

4 Select the **Interior boundaries** check box.

5 Define boundary settings for the glass boundaries according to the following table:

SETTINGS	BOUNDARIES 8, 9, 12–14, 19, 21, 22–24	BOUNDARIES 11, 25
Boundary condition	Heat source/sink	Heat flux
h	0	`h0`
T_{inf}	`25[degC]`	`25[degC]`
Radiation type	Surface-to-surface	Surface-to-ambient
ε	`eps_glass`	`eps_glass`
T_{amb}	T	`25[degC]`

6 Select Boundaries 15–18 from the list (the filament boundaries).

7 From the **Boundary condition** list, select **Heat source/sink**.

8 From the **Library coefficient** list, select **Tungsten**.

9 Set the **Radiation type** to **Surface-to-surface**.

10 In the $\mathbf{T_{inf}}$ edit field type `25[degC]`, and in the $\mathbf{T_{amb}}$ edit field type `T`.

11 Click **OK**.

This completes the setup of the heat-transfer problem. Next, edit the material properties for argon to model it as an ideal gas.

1 From the **Options** menu, select **Materials/Coefficients Library**.
2 In the **Materials** area, select **Model (2)>Argon (mat2)**.
3 Select the **Hide undefined properties** check box.
4 In the **Value/Expression** edit field for **rho**, substitute the existing expression with `rho_a`.
5 Click **OK**.

   Because you have altered the material properties for tungsten, you must reload the material on the appropriate subdomain.

6 Open the **Subdomain Settings** dialog box and select Subdomain 2.
7 From the **Library material** list, select the blank entry to alter the settings in the material-property edit fields.
8 From the **Library material** list, select **Argon** to obtain the updated material properties.
9 Click **OK**.

PHYSICS SETTINGS—WEAKLY COMPRESSIBLE NAVIER-STOKES

1 From the **Model Tree**, select **Weakly Compressible Navier-Stokes (chns)**.
2 Open the **Subdomain Settings** dialog box by double-clicking **Subdomain Settings** under **Weakly Compressible Navier-Stokes (chns)** in the **Model Tree**.
3 Select Subdomains 1 and 3.
4 Deactivate the two solid domains by clearing the **Active in this domain** check box.
5 Select Subdomain 2.
6 From the **Library material** list, select **Argon**.
7 In the **Volume force, z dir.** edit field, type `-rho_chns*g`.
8 Click the **Init** tab.
9 In the **Pressure** edit field, type `p0`.
10 Click **OK**.
11 Open the **Boundary Settings** dialog box.
12 Select Boundaries 2–4.
13 In the **Boundary type** list, select **Symmetry boundary**.
14 From the **Boundary condition** list, select **Axial symmetry**.
15 Click **OK**.

This completes the setup for the momentum transport. You do not need to fix the pressure at a point because it is implicitly defined through the ideal gas law.

MESH GENERATION

1. Open the **Free Mesh Parameters** dialog box.
2. Select Boundaries 12 and 24.
3. Click the **Distribution** tab.
4. Select the **Constrained edge element distribution** check box.
5. In the **Number of edge elements** edit field, enter `50`.
6. Select Boundaries 15–18 (the filament boundary).
7. Select the **Constrained edge element distribution** check box.
8. In the **Number of edge elements** edit field, enter `2`.
9. Click the **Remesh** button.
10. Click **OK**.

SOLVING THE MODEL

1. Open the **Solver Parameters** dialog box.
2. In the **Times** edit field, type `0:0.1:1 1.5:0.5:20 21:3:300`.
3. In the **Relative tolerance** field, type `1e-5`.
4. In the **Absolute tolerance** edit field, type `T 1e-4 J 1e-4 u 1e-4 v 1e-4 p 1e-3`.
   The default absolute tolerance value is 10^{-4} for all dependent variables. To make this model converge quickly, you increase the tolerance for the pressure by a factor 10.
5. From the **Linear system solver** list, select **Direct (UMFPACK)**.
6. Click the **Time Stepping** tab.
7. In the **Time step** edit field, enter `if(t<10,10,100)*dt_cfl_g1_chns`. This means that the time step is 10 times the CFL number for the first 10 seconds, and then 100 times the CFL number.
8. Click **OK**.
9. Because of the small dimensions in the model, also adjust the viscous velocity factor. To do so, choose **Scalar Variables** from the **Physics** menu.
10. In the **Application Scalar Variables** dialog box, enter `0.1` in the **Viscous velocity factor** edit field.
11. Click **OK**.
12. Click the **Solve** button on the Main toolbar to solve the model.

POSTPROCESSING AND VISUALIZATION

To produce the series of pictures in Figure 2-29, proceed as follows:

1. Click the **Plot Parameters** button on the Main toolbar.
2. On the **General** page, select **2** from the **Solution at time** list.
3. Click the **Surface** tab.
4. From the **Predefined quantities** list, select **General Heat Transfer (htgh)>Temperature**.
5. Click the **Range** button.
6. Clear the **Auto** check box. Enter `298` and `2010` in the **Min** and **Max** edit fields, respectively. Click **OK**.
7. From the **Colormap** list, select **hot**.
8. Click **Apply** to generate the leftmost plot.

You can produce the middle and rightmost plots in Figure 2-29 by repeating the above steps but selecting **6** and **10**, respectively, from the **Solution at time** list.

The three snapshots in Figure 2-30 are produced following these steps:

1. On the **General** page, select **2** from the **Solution at time** list.
2. On the **Surface** page, select **Weakly Compressible Navier-Stokes (chns)>Velocity field**.
3. Click the **Range** button.
4. Enter `0` and `0.24` in the **Min** and **Max** edit fields, respectively. Click **OK**.
5. From the **Colormap** list, select **jet**.
6. Click **Apply** to generate the leftmost plot.

The two other plots in the same picture are created by selecting **6** and **10** from the **Solution at time** list respectively.

Figure 2-31 shows the temperature distribution in the bulb after 5 minutes. To produce this plot, do the following:

1. On the **General** page, select **300** from the **Solution at time** list.
2. Click the **Surface** tab.
3. From the **Predefined quantities** list, select **General Heat Transfer (htgh)>Temperature**.
4. From the **Colormap** list, select **hot**.
5. Click **OK**.

Figure 2-32 shows how the temperature evolves over time at a point on the outer boundary of the bulb at the same vertical level as the filament. To produce this plot, follow these instructions:

1 From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box.
2 Click the **Point** tab.
3 From the **Predefined quantities** list, select **General Heat Transfer (htgh)>Temperature**.
4 From the **Point selection** list, select Point 24.
5 Click **Apply**.

Figure 2-33 shows the net radiative heat flux on the outer boundary of the bulb after 5 minutes. You can create this plot with the following instructions:

1 Click the **General** tab.
2 Select **300** from the **Solutions to use** list.
3 Click the **Line/Extrusion** tab.
4 From the **Predefined quantities** list, select **General Heat Transfer (htgh)>Radiative flux**.
5 From the **Boundary selection** list, select **11** and **25**.
6 In the **x-axis data** area, first click the lower option button and then click the **Expression** button to open the **X-Axis Data** dialog box.
7 From the **Predefined quantities** list, select **Geometry and Mesh>z-coordinate** and from the **Unit** list select **cm**. Click **OK**.
8 Click **OK**.

Heating Circuit

Introduction

Small heating circuits find use in many applications. For example, in manufacturing processes they heat up reactive fluids. Figure 2-34 illustrates a typical heating device for this application. The device consists of an electrically resistive layer deposited on a glass plate. The layer causes Joule heating when a voltage is applied to the circuit. The layer's properties determine the amount of heat produced.

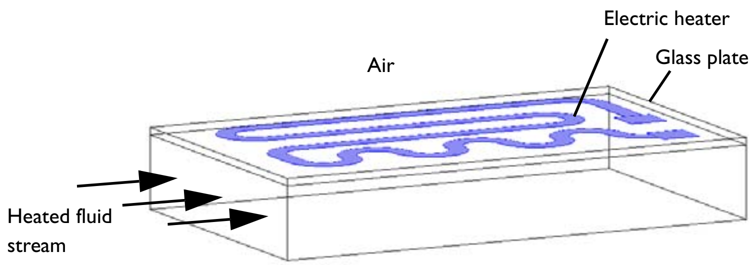


Figure 2-34: Geometry of a heating device.

In this particular application, you must observe three important design considerations:

- Noninvasive heating
- Minimal deflection of the heating device
- Avoidance of overheating the process fluid

The heater must also work without failure. You achieve the first and second requirements by inserting a glass plate between the heating circuit and the fluid; it acts as a conducting separator. Glass is an ideal material for both these purposes because it is nonreactive and has a low thermal-expansion coefficient.

You must also avoid overheating due to the risk of self-ignition of the reactive fluid stream. Ignition is also the main reason for separating the electrical circuit from direct contact with the fluid. The heating device is tailored for each application, making virtual prototyping very important for manufacturers.

For heating circuits in general, detachment of the resistive layer often determines the failure rate. This is caused by excessive thermally induced interfacial stresses. Once the layer has detached, it gets locally overheated, which accelerates the detachment. Finally, in the worst case, the circuit might overheat and burn. From this perspective, it is also important to study the interfacial tension due to the different thermal-expansion coefficients of the resistive layer and the substrate as well as the differences in temperature. The geometric shape of the layer is a key parameter to design circuits that function properly. You can investigate all of the above-mentioned aspects by modeling the circuit.

This multiphysics example simulates the electrical heat generation, the heat transfer, and the mechanical stresses and deformations of a heating circuit device. The model uses the General Heat Transfer application mode of the Heat Transfer module in combination with the Shell, Conductive Media DC application mode from the AC/DC Module and the Solid, Stress-Strain and Shell application modes from the Structural Mechanics Module.

Note: This model requires the AC/DC Module, the Heat Transfer Module, and the Structural Mechanics Module.

Model Definition

Figure 2-35 shows a drawing of the modeled heating circuit.

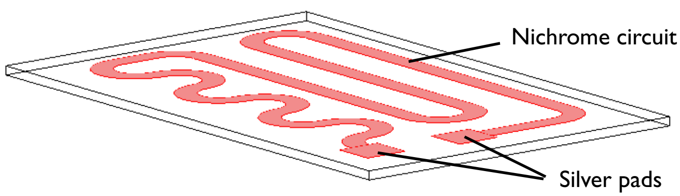


Figure 2-35: Drawing of the heating circuit deposited on a glass plate.

The device consists of a serpentine-shaped Nichrome resistive layer, 10 µm thick and 5 mm wide, deposited on a glass plate. At each end, it has a silver contact pad measuring 10 mm × 10 mm × 10 µm. When the circuit is in use, the deposited side of the glass plate is in contact with surrounding air, and the back side is in contact with the heated fluid. Assume that the edges of the glass plate are thermally insulated.

Table 2-4 gives the resistor's dimensions.

TABLE 2-4: DIMENSIONS

OBJECT	DIMENSION	SIZE
glass plate	length	130 mm
	width	80 mm
	thickness	2 mm
pads and circuit	thickness	10 µm

During operation the resistive layer produces heat. Model the electrically generated heat using the Shell, Conductive Media DC application mode from the AC/DC Module. The governing equation is

$$\nabla_t \cdot (-d\sigma\nabla_t V) = 0$$

where d is the thin layer's thickness (m), σ is the electric conductivity (S/m), V is the electric potential (V), and ∇_t denotes the gradient operator in the tangential directions. An actual applies 12 V to the pads. In the model you achieve this effect by setting the potential at one edge of the first pad to 12 V and that of one edge of the other pad to 0 V.

To model the heat transfer in the thin conducting layer, use the Highly Conductive Layer feature of the General Heat Transfer application mode. It is then not necessary to add a separate application mode for it.

The heat power per unit area (measured in W/m^2) produced inside the thin layer is given by

$$q_{\mathrm{prod}} = dQ_{\mathrm{DC}} \qquad (2\text{-}1)$$

where $Q_{\mathrm{DC}} = \mathbf{J}\cdot\mathbf{E} = \sigma|\nabla_t V|^2$ (W/m^3) is the power density. The generated heat appears as an inward heat flux at the surface of the glass plate.

At steady state, the resistive layer dissipates the heat it generates in two ways: on its up side to the surrounding air (at 293 K), and on its down side to the glass plate. The glass plate is similarly cooled in two ways: on its circuit side by air, and on its back side by a process fluid (353 K). You model the heat fluxes to the surroundings using heat transfer film coefficients, h. For the heat transfer to air, $h = 5$ W/(m^2·K), representing natural convection. On the glass plate's back side, $h = 20$ W/(m^2·K), representing convective heat transfer to the fluid. The sides of the glass plate are insulated.

The resulting heat transfer equation for the device, together with the boundary condition used to describe the heat fluxes at the front and back sides, is

$$\nabla \cdot (-k\nabla T) = 0$$

$$-\mathbf{n} \cdot (-k\nabla T) = q_0 + h(T_{\text{inf}} - T) - \nabla_t \cdot (-d_s k_s \nabla_t T)$$

where $\mathbf{n}$ is the normal vector of the boundary, k is the thermal conductivity (W/(m·K)), h is the heat transfer film coefficient (W/(m^2·K)), and T_{inf} is the temperature (K) of the surrounding medium. The last term on the right-hand side represents the additional flux given by the thin conducting layer, and the constant k_s is the thermal conductivity in the layer (W/(m·K)). This term is only present on the boundaries where the layer is present. Similarly, the inward heat flux, q_0, is equal to q_{prod} (see Equation 2-1) at the layer but vanishes elsewhere.

The model simulates thermal expansion using static structural-mechanics analyses. It uses the Solid, Stress-Strain application mode for the glass plate, and the Shell application mode for the circuit layer. The equations of these two application modes are described in the *Structural Mechanics Module User's Guide*. The stresses are set to zero at 293 K. You determine the boundary conditions for the Solid, Stress-Strain application mode by fixing one corner with respect to x-, y-, and z-displacements and rotation.

Table 2-5 summarizes the material properties used in the model.

TABLE 2-5: MATERIAL PROPERTIES

MATERIAL	E [GPa]	ν	α [ppm]	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silver	83	0.37	18.9	420	10500	230
Nichrome	213	0.33	10.0	15	9000	20
Glass	73.1	0.17	55	1.38	2203	703

Results and Discussion

Figure 2-36 shows the heat that the resistive layer generates.

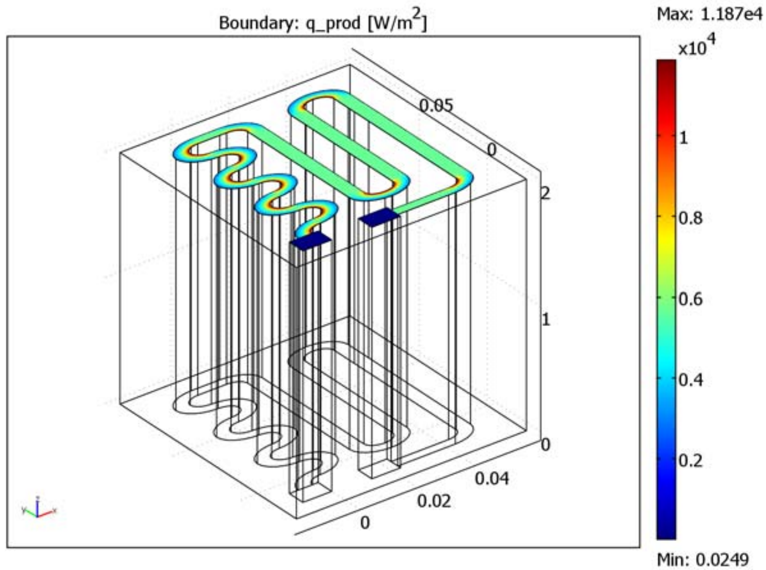


Figure 2-36: Stationary heat generation in the resistive layer when 12 V is applied.

The highest heating power arises in the inner corners of the curves due to the higher current density at these spots. The total generated heat, as calculated by integration, is approximately 13.8 W.

Figure 2-37 shows the temperature of the resistive layer and the glass plate at steady state.

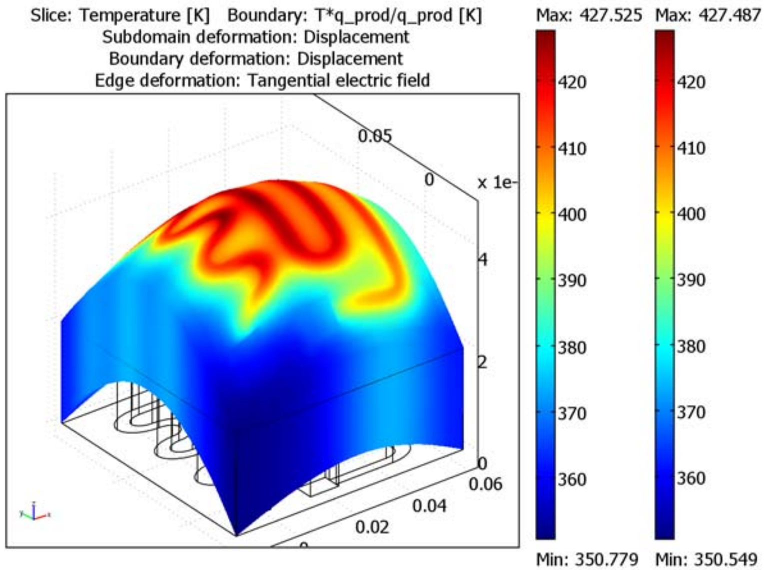


Figure 2-37: Temperature distribution in the heating device at steady state.

The highest temperature is approximately 430 K, and it appears in the central section of the circuit layer. It is interesting to see that the differences in temperature between the fluid side and the circuit side of the glass plate are quite small because the plate is very thin. Using boundary integration, the integral heat flux on the fluid side evaluates to approximately 8.5 W. This means that the device transfers the majority of the heat it generates—8.5 W out of 13.8 W—to the fluid, which is good from a design perspective, although the thermal resistance of the glass plate results in some losses.

The temperature rise also induces thermal stresses due the materials' different coefficients of thermal expansion. As a result, mechanical stresses and deformations arise in the layer and in the glass plate. Figure 2-38 shows the effective stress distribution in the device and the resulting deformations. During operation, the glass plate bends towards the air side.

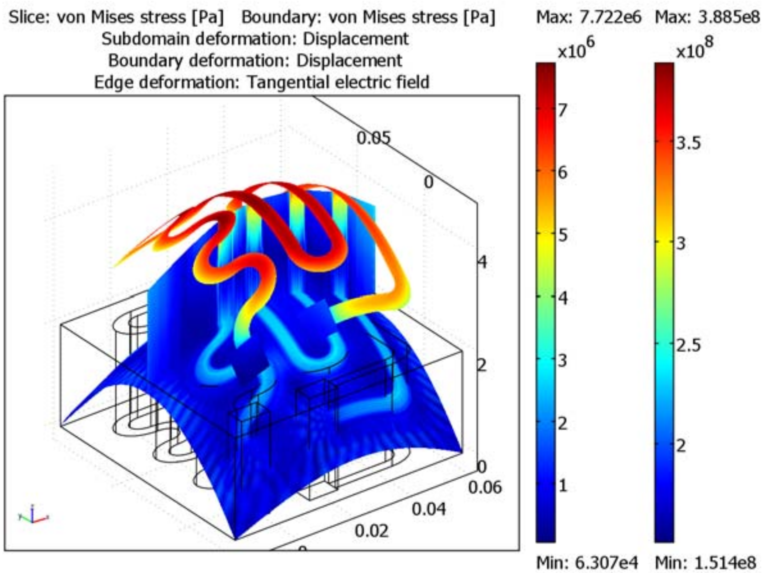


Figure 2-38: The thermally induced von Mises effective stress plotted with the deformation.

The highest effective stress, approximately 7 MPa, appears in the corners of the silver pads. The yield stress for high quality glass is roughly 250 MPa, and for Nichrome it is 360 MPa. This means that the individual objects remain structurally intact for the simulated heating power loads.

You must also consider stresses in the interface between the resistive layer and the glass plate. Assume that the yield stress of the surface adhesion in the interface is in the region of 50 MPa—a value significantly lower than the yield stresses of the other materials in the device. If the effective stress increases above this value, the resistive layer will locally detach from the glass. Once it has detached, heat transfer is locally impeded, which can lead to overheating of the resistive layer and eventually cause the device to fail.

Figure 2-39 displays the effective forces acting on the adhesive layer during heater operation. As the figure shows, the device experiences a maximum interfacial stress that is an order of magnitude smaller than the yield stress. This means that the device will be OK in terms of adhesive stress.

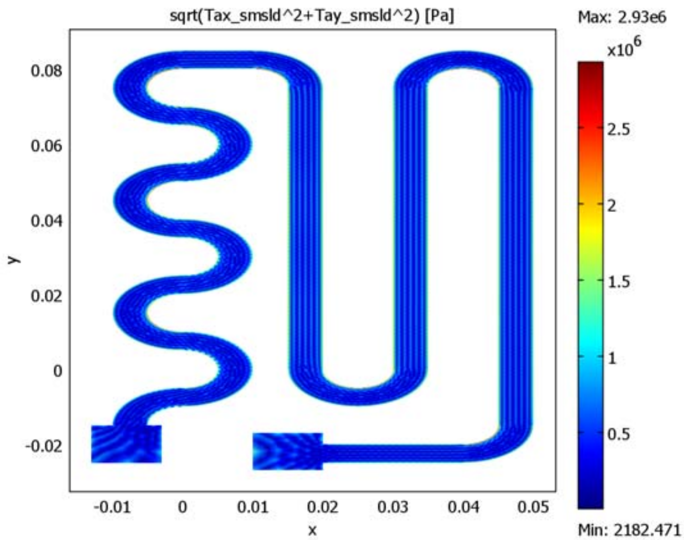


Figure 2-39: The effective forces in the interface between the resistive layer and the glass plate.

Finally study the device's deflections, depicted in Figure 2-40.

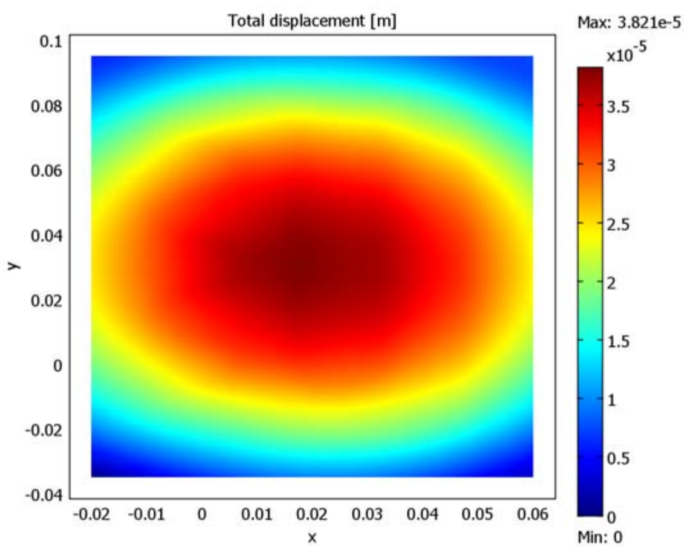


Figure 2-40: Total displacement on the fluid side of the glass plate.

The maximum displacement, located at the center of the plate, is approximately 30 µm. For high-precision applications, such as semiconductor processing, this might be a significant value that limits the device's operating temperature.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/heating_circuit`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Open the **Model Navigator**.
2. In the **Space dimension** list select **3D**. Click the **Multiphysics** button.
3. From the list of application modes select **AC/DC Module>Statics, Electric>Shell, Conductive Media DC**. In the **Application mode name** edit field type `DC`, then click **Add**.
4. From the list of application modes select **Heat Transfer Module>General Heat Transfer**, then click **Add**.
5. Similarly add two more application modes: **Structural Mechanics Module>Solid, Stress-Strain** and **Structural Mechanics Module>Shell**. When done, click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu select **Constants**. Define the following names, expressions, and descriptions (the descriptions are optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
V_in	12[V]	Input voltage
d_layer	10[um]	Layer thickness
sigma_Silver	6.3e7[S/m]	Electric conductivity of silver
sigma_Nichrome	9.3e5[S/m]	Electric conductivity of Nichrome
T_ref	293[K]	Reference temperature
T_air	T_ref	Air temperature
h_air	5[W/(m^2*K)]	Heat transfer film coefficient, air
T_fluid	353[K]	Fluid temperature
h_fluid	20[W/(m^2*K)]	Heat transfer film coefficient, fluid

GEOMETRY MODELING

1 Create a 2D work plane at $z = 0$ by first choosing **Draw>Work-Plane Settings** and then clicking **OK** in the dialog box that appears to accept the default settings.

2 Create nine rectangles. Open the **Draw>Specify Objects>Rectangle** dialog box, and for each one enter the appropriate data from this table:

RECTANGLE	WIDTH	HEIGHT	X-BASE	Y-BASE
R1	0.08	0.13	-0.02	-0.035
R2	0.01	0.02	-0.01	-0.01
R3	0.005	0.075	0.015	0
R4	0.01	0.005	0	0.08
R5	0.01	0.01	0	0.075
R6	0.005	0.015	0.045	0.06
R7	0.02	0.005	0.02	-0.025
R8	0.01	0.01	0.01	-0.027
R9	0.01	0.01	-0.013	-0.025

3 Click the **Zoom Extents** button on the Main toolbar.

4 Create a circle with the menu item **Draw>Specify Objects>Circle**. In the **Radius** edit field type `0.01`, then click **OK**.

5 Create another circle the same way bit with a **Radius** of `0.005`.

6 Click the **Create Composite Object** button on the Draw toolbar. In the **Set formula** edit field type `C1-C2-R2`, then click **OK**. This step generates the composite object CO1.

7 Select CO1 by clicking on it, click the **Mirror** button on the Draw toolbar, then click **OK**. This step creates a mirror copy of CO1 called CO2.

8 Select CO2 and then click the **Move** button on the Draw toolbar. Go to the **Displacement** area, and in the **y** edit field type `0.015`. Click **OK**.

9 Select both CO1 and CO2 by pressing Ctrl while clicking on the objects. Copy both by pressing Ctrl+C.

10 Paste twice by pressing Ctrl+V, specifying the displacement, and clicking **OK**. For the first copy specify the y-displacement as `0.030`, and for the second specify `0.060`.

11 Select CO1, copy it, and paste it with zero displacement.

12 Click the **Rotate** button on the Draw toolbar. In the **Rotation angle** edit field type `-90`, then click **OK**.

13 Click the **Move** button on the Draw toolbar. In the **x** edit field type `0.025`, then click **OK**.

14 Click the **Mirror** button on the Draw toolbar. In the **Normal vector** edit field for **x** type `0`, and in the **y** edit field type `1`. Click **OK**.

15 Click the **Move** button on the Draw toolbar. In the **x** edit field type `-0.015`, and in the **y** edit field type `0.075`. Click **OK**.

16 Click the **Create Composite Object** button on the Draw toolbar. In the **Set formula** edit field type `CO8-R5`, then click **OK**. This step generates composite object CO9.

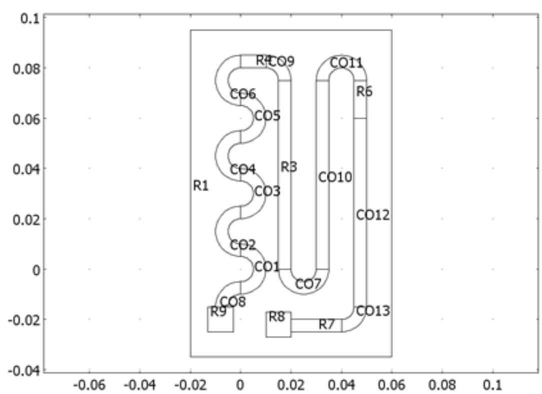
17 Copy and paste CO9 with *x*- and *y*-displacements of `-0.02` and `-0.08`, respectively.

18 Click the **Rotate** button on the Draw toolbar. In the α edit field in the **Rotation angle** area type `90`, and in the **y** edit field in the **Center point** area type `-0.005`. Click **OK**.

19 Select objects CO7 and R3. Click the **Mirror** button on the Draw toolbar. In the **Normal vector** edit field for **x** type `0`, and in the **y** edit field type `1`. Click **OK**.

20 Click the **Move** button on the Draw toolbar. Specify the displacement by typing `0.015` in the **x** edit field and `0.075` in the **y** edit field, then click **OK**. These steps generate composite objects CO10 and CO11.

21 Select objects CO9 and R3. Repeat the procedures in the previous two steps, using the **Move** dialog box with values for the *x*- and *y*-displacements of `0.03` and `0.06`. The following figure shows the geometry after this step.



22 Select all objects except the glass plate (R1) and the silver tabs (R8 and R9). Click the **Create Composite Object** button on the Draw toolbar. Clear the **Keep interior boundaries** check box, then click **OK**. This step generates composite object CO14.

23 Select CO14, R8, and R9, then click the **Coerce to Solid** button on the Draw toolbar.

MESH GENERATION

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box.

2 On the **Global** page go to the **Predefined mesh sizes** list and select **Coarse**.

3 On the **Subdomain** page select Subdomain 3, then in the **Maximum element size** edit field type `2e-3`.

4 Click the **Remesh** button, then click **OK**.

5 From the **Mesh** menu open the **Extrude Mesh** dialog box. On the **Geometry** page find the **Distance** edit field and type `1e-3`. From the **Extrude to geometry** list select **Geom1**.

6 Click the **Mesh** tab. In the **Number of element layers** edit field type `2`, then click **OK**.

7 Double-click the **EQUAL** button on the status bar at the bottom of the user interface, then click the **Zoom Extents** button on the Main toolbar to expand the geometry's z-axis.

PHYSICS SETTINGS

1 From the **Options** menu open the **Materials/Coefficients Library** dialog box.

2 Set up the materials silver and NiChrome. To do so, click **New**, then enter the settings from the following table in the corresponding edit fields. When done, click **OK**.

NAME	C	E	alpha	k	nu	rho
Silver	230	83e9	18.9e-6	420	0.37	10500
Nichrome	230	213e9	10e-6	15	0.33	9000

3 Choose **Options>Expressions>Scalar Expressions**. In the **Name** edit field type `q_prod`, and in the **Expression** edit field type `d_layer*Q_DC`. Enter `Heat power per unit area inside thin layer` in the **Description** edit field (optional). Click **OK**.

Boundary Settings—Shell, Conductive Media DC (DC)

1 From the **Multiphysics** menu select **Shell, Conductive Media DC (DC)**.

2 From the **Physics** menu select **Boundary Settings**.

3 Select all the boundaries, then clear the **Active in this domain** check box.

4 Select Boundary 14, then click the **Active in this domain** check box. In the **Electric conductivity** edit field for σ type `sigma_Nichrome`, and in the **Thickness** edit field type `d_layer`.

5 Repeat the previous step for Boundaries 9 and 47 but in the **Electric conductivity** edit field type `sigma_Silver`. Click **OK**.

6 From the **Physics** menu select **Edge Settings**.

7 Select Edge 13. In the **Boundary condition** list select **Electric potential**, then in the **Electric potential** edit field type `V_in`.

8 Select Edge 109. In the **Boundary condition** list select **Ground**, then click **OK**.

Subdomain Settings—General Heat Transfer

1 From the **Multiphysics** menu select **General Heat Transfer (htgh)**.

2 From the **Physics** menu open the **Subdomain Settings** dialog box, then select all the subdomains.

3 Go to the **General** page. Click the **Load** button. From the **Materials** list select **Basic Material Properties>Silica glass**, then click **OK**.

4 Go to the **Init** page, and in the **$T(t_0)$** edit field type `T_ref`. Click **OK**.

Boundary Conditions—General Heat Transfer

1 From the **Physics** menu open the **Boundary Settings** dialog box. Select Boundaries 9, 14, and 47.

2 Click the **Highly Conductive Layer** tab. Select the **Enable heat transfer in highly conductive layer** check box, then in the **d_s** edit field type `d_layer`.

3 Select Boundary 14. In the **Library material** list select **Nichrome**.

4 Similarly, for Boundaries 9 and 47 select **Silver**.

5 Click the **Boundary Condition** tab. Select Boundaries 9, 14, and 47. In the **Boundary condition** list select **Heat flux**. In the **q_0** edit field type `q_prod`, in the **h** edit field type `h_air`, and in the **T_{inf}** edit field type `T_air`.

6 Repeat the settings in the previous step for Boundary 4 but without specifying **q_0**.

7 Select Boundaries 3, 8, 13, and 46. In the **Boundary condition** list select **Heat flux**. In the **h** edit field type `h_fluid`, and in the **T_{inf}** edit field type `T_fluid`. Click **OK**.

Subdomain Settings—Solid, Stress-Strain

1 From the **Multiphysics** menu select **Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu open the **Subdomain Settings** dialog box, then select all the subdomains.

3 Go to the **Material** page. In **Library material** list select **Silica glass**.

4 Click the **Load** tab. Select the **Include thermal expansion** check box. In the **Temp** edit field type `T` and in the **Tempref** edit field type `T_ref`.

5 Go to the **Element** page. In the **Predefined elements** list select **Lagrange - Linear**. Click **OK**.

Point Settings—Solid, Stress-Strain

1. From the **Physics** menu open the **Point Settings** dialog box.
2. Select Point 1. Select the check boxes next to $\mathbf{R_x}$, $\mathbf{R_y}$, and $\mathbf{R_z}$.
3. Select Point 3, then select the $\mathbf{R_z}$ check box.
4. Select Point 125, then select the $\mathbf{R_y}$ and $\mathbf{R_z}$ check boxes. Click **OK**.

Boundary Settings—Shell

1. From the **Multiphysics** menu select **Shell (smsh)**.
2. From the **Physics** menu open the **Boundary Settings** dialog box.
3. Select all the boundaries, then clear the **Active in this domain** check box.
4. Select Boundaries 9, 14, and 47. Select the **Active in this domain** check box.
5. In the **thickness** edit field type `d_layer`.
6. Go to the **Load** page. Select the **Include thermal expansion** check box. In the **Temp** edit field type `T` and in the **Tempref** edit field type `T_ref`.
7. Click the **Material** tab. Select Boundaries 9 and 47. In the **Library material** list select **Silver**.
8. Similarly, for Boundary 14 select **Nichrome**. Click **OK**.

COMPUTING THE SOLUTION

This model is best solved using a script. Follow these steps to create the script and solve the model.

1. From the **Solve** menu open the **Solver Manager**.
2. On the **Solve For** page select **Geom1 (3D)>Shell, Conductive Media DC (DC)** and **Geom1 (3D)>General Heat Transfer (htgh)**, then click **Apply**.
3. Go to the **Script** page. Select the **Solve using a script** check box. Then click the **Add Current Solver Settings** button to generate the first half of the script.
4. From the **Solve** menu, choose **Solver Parameters**.
5. In the **Linear system solver** list select **Direct (SPOOLES)** to use this solver's ability to utilize the symmetric system matrices. Click **OK**.
6. Return to the **Solver Manager**. Go to the **Initial Value** page, then to the **Initial value** area, and click the **Current solution** option button.
7. Go to the **Solve For** page. Select **Geom1>Solid, Stress-Strain (smsld)** and **Geom1>Shell (smsh)**. Click **Apply**.

8 On the **Script** page click the **Add Current Solver Settings** button to generate the second half of the script. Click **OK** to close the **Solver Manager**.

9 Finally, click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

Generate Figure 2-36 as follows:

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box.

2 On the **General** page clear the **Slice** check box, then select the **Boundary** check box.

3 Go to the **Boundary** page. In the **Expression** edit field type `q_prod`. Click **OK**.

4 Click the **Zoom Extents** button on the Main toolbar.

To calculate the total heat generated in the circuit, follow these steps:

1 From the **Postprocessing** menu open the **Boundary Integration** dialog box.

2 Select Boundaries 9, 14, and 47. In the **Expression** edit field type `q_prod`. Click **OK**.

The calculated value, roughly 13.8 W, appears in the message log at the bottom of the graphical user interface.

Generate Figure 2-37 by executing these instructions:

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box. On the **General** page go to the **Plot type** area and select the **Slice**, **Boundary**, and **Deformed shape** check boxes.

2 Go to the **Slice** page, and in the **Predefined quantities** list select **General Heat Transfer (htgh)>Temperature**. In the **Slice positioning** area find the **x levels** edit field and type `0`, and in the **y levels** edit field type `1`.

3 Click the option button for **Vector with coordinates** associated with **z levels**, then in the corresponding edit field type `0`.

4 On the **Boundary** page find the **Expression** edit field and type `T*q_prod/q_prod`.

The use of `q_prod/q_prod` makes the expression for the temperature valid on the resistive layer boundary only, which is the desired effect.

5 On the **Deform** page, go to the **Deformation data** area and click the **Subdomain Data** tab. In the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>Displacement**.

6 While still in the **Deformation data** area, click the **Boundary Data** tab. In the **Predefined quantities** list select **Shell (smsh)>Displacement**. Click **Apply**.

Calculate the total heat flux to the fluid in the following way:

1 From the **Postprocessing** menu open the **Boundary Integration** dialog box.

2 Select Boundaries 3, 8, 13, and 46. In the **Expression** edit field type `h_fluid*(T-T_fluid)`, then click **OK**.

   A value for the total heat flux of approximately 8.47 W appears in the message log.

To generate Figure 2-38 follow these steps:

1 While still in the **Plot Parameters** dialog box, go to the **Slice** page. In the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>von Mises stress**.

2 Click the **Boundary** tab. In the **Predefined quantities** list select **Shell (smsh)>von Mises stress**. Click **Apply**.

Figure 2-39 is obtained by executing the following instructions:

1 From the **Postprocessing** menu select **Domain Plot Parameters**.

2 On the **Surface** page select Boundaries 9, 14, and 47. In the **Expression** edit field type `sqrt(Tax_smsld^2+Tay_smsld^2)`. Click **Apply**.

   This gives a plot of the norm of the surface traction vector (N/m^2) in the surface plane,

$$\begin{bmatrix} \mathrm{Ta}_x \\ \mathrm{Ta}_y \end{bmatrix} = \begin{bmatrix} \sigma_x & \tau_{xy} & \tau_{xz} \\ \tau_{yx} & \sigma_y & \tau_{yz} \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ 1 \end{bmatrix} = \begin{bmatrix} \tau_{xz} \\ \tau_{yz} \end{bmatrix}$$

Finally, to obtain Figure 2-40, proceed as follows:

1 Still on the **Surface** page of the **Domain Plot Parameters** dialog box, select Boundaries 3, 8, 13, and 46.

2 In the **Predefined quantities** list select **Solid, Stress-Strain (smsld)>Total displacement**, then click **OK**.

Microchannel Heat Sink

Introduction

This example models a microchannel heat sink mounted on an active electronic component. The model geometry is based on a paper by B.C. Pal and others (Ref. 1) as well as another from S.P. Jang and others (Ref. 2).

Thermal management has become a critical aspect of today's electronic systems, which often include many high-performance circuits that dissipate large amounts of heat. Many of these components require efficient cooling to prevent overheating. Some of these components, such as processors, require a heat sink with cooling fins that are exposed to forced air from a fan. This discussion develops the model of an aluminum microchannel heat sink whose manifolds work as flow dividers to improve its cooling performance (see Figure 2-41).

This case examines the temperature field in the air, in the aluminum, and in the heat source. The air transports heat by convection and conduction. Because the geometry is fairly complicated, it is not possible to use an analytical expression for the velocity profile, so you must also model the fluid flow and couple it to the heat equation. The aluminum heat sink transports thermal energy by pure conduction. Finally, to approximate the electronic component that requires cooling, the model uses a rectangular block with a given volume heat source.

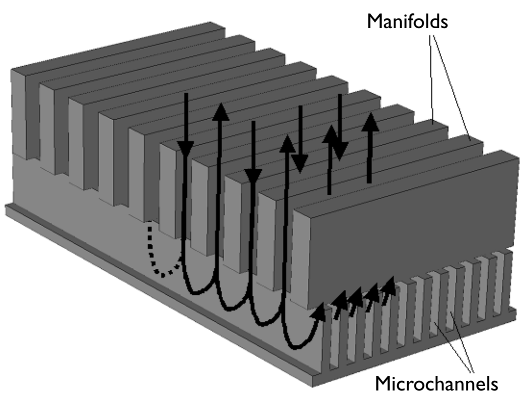


Figure 2-41: Microchannel heat sink with manifolds.

Model Definition

The model geometry consists of three subdomains: the electronic component, the aluminum heat sink, and the cooling air. Because of symmetry, it is sufficient to model just a small element of the entire geometry as shown in Figure 2-42.

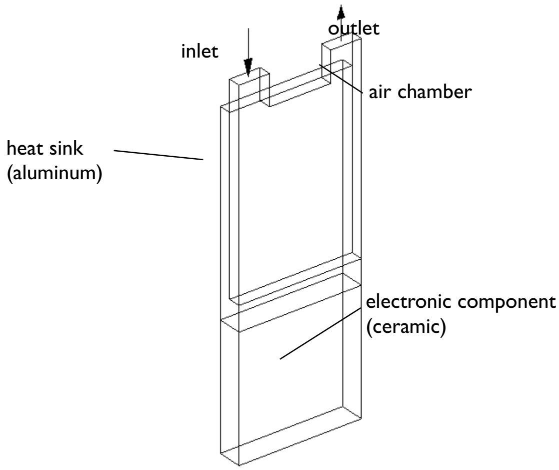


Figure 2-42: Symmetry element for modeling the heat sink and the heat source.

In particular, the surfaces labeled "inlet" and "outlet" in the figure each constitutes a quarter of the actual inlet and outlet, respectively.

This simulation employs the General Heat Transfer and the Weakly Compressible Navier-Stokes application modes for stationary analysis. The first mode models the temperature field in the entire geometry; the second one serves only to model the airflow.

Both the component and the heat sink transport heat by pure conduction as described by the conductive heat equation

$$\nabla \cdot (-k\nabla T) = Q$$

where k (W/(m·K)) is the thermal conductivity, Q (W/m^3) is the heat source, and T (K) denotes the temperature. The model's heat source relates to the component's output power, and Q equals zero for the heat sink because that device has no heat sources.

The temperature field in the air is governed by the heat equation for conduction and convection

$$\rho C_p \mathbf{u} \cdot \nabla T - \nabla \cdot (k \nabla T) = 0$$

where k refers to the thermal conductivity (W/(m·K)), ρ is the density (kg/m^3) and C_p denotes the specific heat capacity (J/(kg·K)) for air. You obtain the velocity vector $\mathbf{u}$ (m/s) from the equations for the airflow as described later in this section.

The boundary conditions for the heat-transfer equations are:

$$T = T_{\text{in}} \qquad \text{at the inlet,}$$

$$\mathbf{n} \cdot (-k \nabla T) = 0 \qquad \text{elsewhere.}$$

The last equation applies equally well at thermally insulated boundaries and at boundaries through which no heat flows because of symmetry. In addition, assume heat-flux continuity on all interior boundaries.

Now use the Weakly Compressible Navier-Stokes equations for the momentum equations and the equation of continuity to describe the air's velocity and pressure field:

$$\rho \mathbf{u} \cdot \nabla \mathbf{u} = \nabla \cdot [-p\mathbf{I} + \eta (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - (2\eta/3)(\nabla \cdot \mathbf{u})\mathbf{I}] + \mathbf{F} \qquad \text{in the air}$$
$$\nabla \cdot (\rho \mathbf{u}) = 0$$

where η denotes the dynamic viscosity (Pa·s), $\mathbf{u}$ is the velocity (m/s), ρ is the fluid density (kg/m^3), p represents pressure (Pa), and $\mathbf{F}$ is the volume force (N/m^3).

The air density depends on the pressure and temperature according to the ideal gas equation

$$\rho = \frac{p}{RT}$$

where R is the mass-based gas constant, equal to 287 J/(kg·K) for air.

You can assume that the volume force, $\mathbf{F}$, is zero because gravitational forces due to changes in density most likely have very little impact on this model.

At the inlet, the fluid enters with a parabolic velocity profile modeling fully developed laminar flow. The mean velocity is approximately 1 m/s, and the air temperature is 293 K. At the outlet the pressure is 10^5 Pa, and heat leaves through convection.

These assumptions lead to the following boundary conditions for the Weakly Compressible Navier-Stokes application mode:

$\mathbf{u} = 0$	at walls
$\mathbf{n} \cdot \mathbf{u} = 0$	at symmetry boundaries
$p = p_0, \quad [\eta(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - (2\eta/3)(\nabla \cdot \mathbf{u})\mathbf{I}]\mathbf{n} = 0$	at the outlet
$\mathbf{u} = (0, 0, w)$	at the inlet

Here, the outlet pressure equals $p_0 = 1.0 \cdot 10^5$ Pa, and the z-component of the inlet velocity is

$$w = -0.3 \cdot 10^{16}(2.5 \cdot 10^{-4} + x)(2.5 \cdot 10^{-4} - x)(1 \cdot 10^{-4} + y)(1 \cdot 10^{-4} - y) \ \ \mathrm{m/s}$$

with x and y expressed in meters. The prefactor is calculated to give an average inflow speed of 1 m/s at the rectangular inlet surface measuring $5 \cdot 10^{-4}$ m $\times$ $2 \cdot 10^{-4}$ m and centered at $x = 0$, $y = 0$. The model geometry covers only the part where $x \geq 0$, $y \geq 0$.

Adding Thermal Contact Resistance

The model described thus far assumes perfect thermal contact at the interface between the heat source and the aluminum heat sink. A more realistic model accounts for the interface's thermal contact resistance. That resistance is an important factor in the design of electronics cooling because it can significantly reduce a heat sink's cooling performance. The following discussion describes how to account for the thermal contact resistance, starting from the initial model.

The analysis is based partly on reference Ref. 3, which presents an analysis of how to calculate the interface resistance for a ceramic package/aluminum heat-sink assembly.

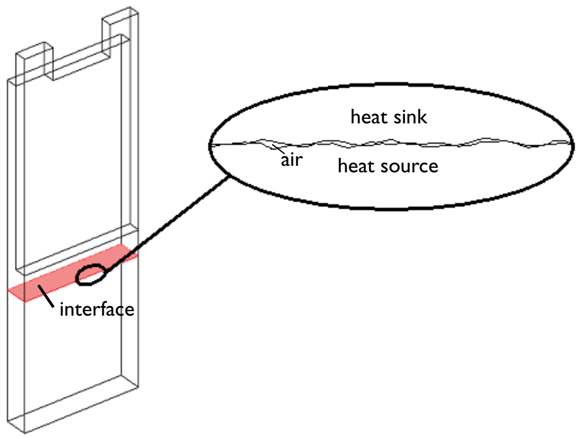


Figure 2-43: The interface between the heat source and the heat sink.

Figure 2-43 shows the interface between the heat source and the heat sink. The surfaces of the heat sink and the ceramic heat source are not in perfect contact because of their roughness; air fills the gaps between the surfaces.

Modeling the interface with the geometry of the rough surfaces would require a very dense mesh. An alternative, more practical, way of modeling the interface is to define a thermal joint conductivity, h_j (W/(m^2·K)), that is representative for the interface.

Using the thermal joint conductivity, the heat flux from the heat source to the heat sink is

$$q_{\text{source} \rightarrow \text{sink}} = h_j(T_{\text{source}} - T_{\text{sink}}) \tag{2-2}$$

where T_{sink} is the temperature of the aluminum heat sink at the interface, and T_{source} is the temperature of the ceramic heat source at the interface. Equation 2-2 states that the difference in temperature across the interface drives the heat flux.

Similarly, the heat flux from the heat sink to the heat source is

$$q_{\text{sink} \rightarrow \text{source}} = h_j(T_{\text{sink}} - T_{\text{source}}) \tag{2-3}$$

Note that $q_{\text{sink} \rightarrow \text{source}}$ has a negative value as long as the source temperature is higher than the sink temperature.

These two heat-flux conditions maintain heat flux continuity through the interface, that is, the heat flux out of the heat source equals the heat flux into the heat sink.

Ref. 3 shows how to calculate the thermal joint conductivity of an interface that is similar to the one in this model; here follows a brief summary.

The thermal joint conductivity, h_j, is defined as the sum of the conductivity through those regions that are in contact and those where there is a gap,

$$h_j = h_c + h_g \tag{2-4}$$

where h_c is the contact conductivity and h_g is the gap conductivity, both measured in $W/(m^2 \cdot K)$. The contact conductivity is determined by the expression

$$h_c = 1.25 k_s \frac{m}{\sigma}\left(\frac{P}{H_c}\right)^{0.95} \tag{2-5}$$

where k_s, m, σ, and H_c are parameters specifying the material and surface characteristics of the surfaces, and P is the contact pressure.

The gap conductance is

$$h_g = \frac{k_g}{Y + M} \tag{2-6}$$

where k_g is the thermal conductivity of the air in the gap, Y is the effective gap thickness, and M is a gas parameter that accounts for rarefaction effects at high temperatures and low pressures.

For this model you apply the value $h_j = 5400\ W/(m^2 \cdot K)$. For more details on how to compute these properties according to Equation 2-4, Equation 2-5, and Equation 2-6, see Ref. 3.

In COMSOL Multiphysics you model thermal contact resistance by applying the Thin thermally resistive layer boundary condition:

$$\begin{aligned} -\mathbf{n}_u \cdot (-k_u \nabla T_u) &= \frac{k_{res}}{d_{res}}(T_d - T_u) \\ -\mathbf{n}_d \cdot (-k_d \nabla T_d) &= \frac{k_{res}}{d_{res}}(T_u - T_d) \end{aligned} \tag{2-7}$$

This boundary condition, known as a *slit boundary condition*, can accommodate a discontinuity in the temperature field across the boundary. The parameters of the boundary condition are the layer thermal conductivity k_{res}, and layer thickness d_{res}. For this model, we only know the factor k_{res}/d_{res}, which is equal to our thermal joint conductivity h_j. We can specify the correct thermal joint conductivity by applying the values $k_{res} = h_j \cdot 1.0$ m and $d_{res} = 1.0$ m.

Slit boundary conditions are only available on assembly pair boundaries, which requires us to set up an assembly geometry to model contact resistance.

Results and Discussion

Figure 2-44 shows the resulting temperature field for the initial model. It indicates that this scheme holds the component's temperature at roughly 337 K. The air temperature increases from 295 K to approximately 337 K on its way from the inlet to the outlet, something you could expect because the air absorbs heat energy from the aluminum. The figure also shows streamlines for the total heat flux. The streamlines show that the heat energy leaves the aluminum and escapes through the outlet.

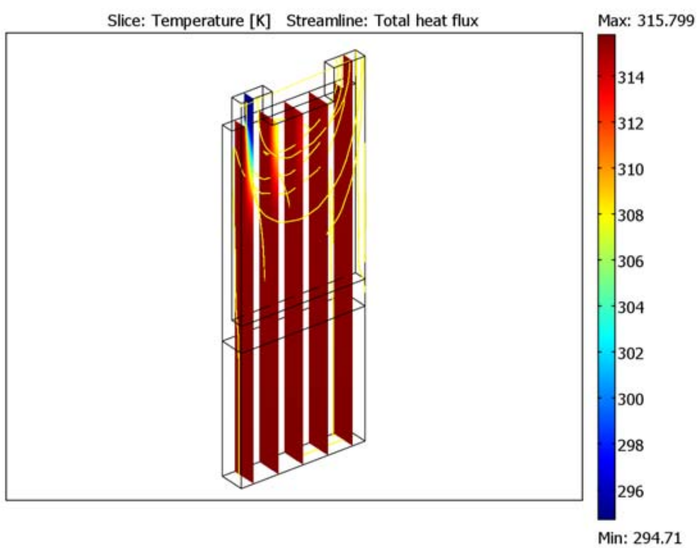


Figure 2-44: Temperature field and heat-flux streamlines.

The velocity field and its streamlines appear in Figure 2-45. Again, as expected, the velocities are highest at the inlet and outlet.

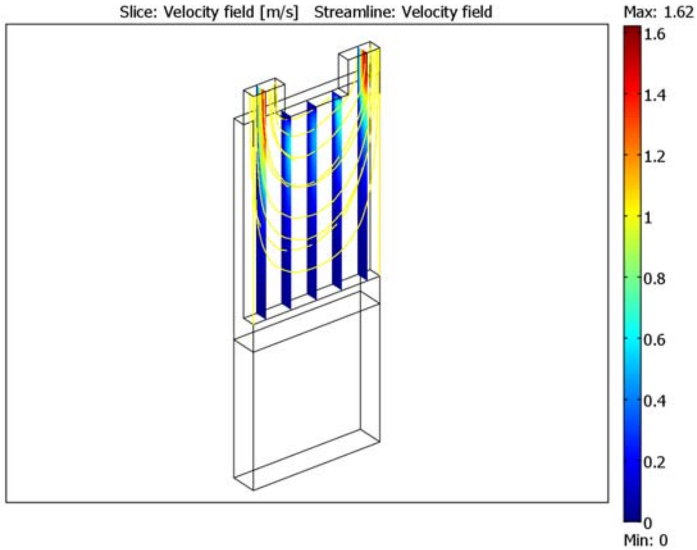


Figure 2-45: Velocity field with velocity streamlines.

Figure 2-46 shows the temperature field of the extended model, which accounts for the thermal contact resistance of the interface between the heat source and sink. A small temperature jump is observed on the interface. The maximum temperature of the component is roughly 1 K higher than the result obtained with the initial model. This confirms that the interface's thermal contact resistance does have an impact on the heat sink's cooling performance, albeit a small one.

Most importantly, the results show that the electronic component does not overheat when operating continuously at the given power.

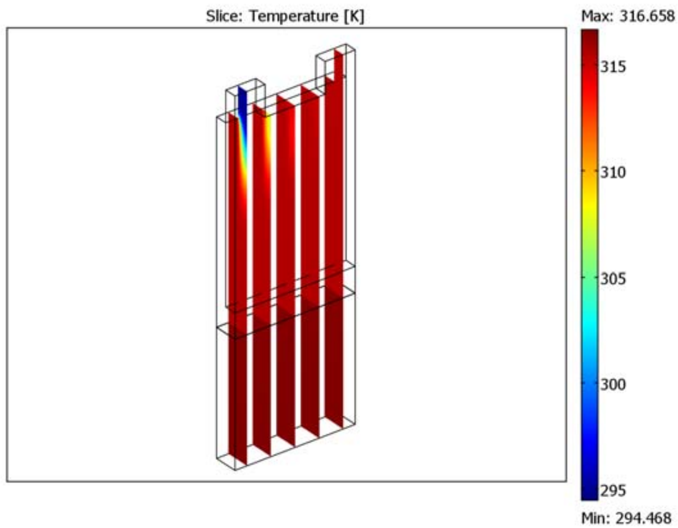


Figure 2-46: Temperature field when accounting for the thermal contact resistance.

References

1. B.C. Pak, W.C. Chun, and B.J. Baek, "Forced Air Cooling by Manifold Microchannel Heat Sinks," EEP, vol. 19.2, *Advances in Electronic Packaging*, ASME, 1997.

2. S.P. Jang, S.J. Kim, and K.W. Paik, "Experimental investigation of thermal characteristics for a microchannel heat sink subject to an impinging jet, using a micro-thermal sensor array," *Sensors and Actuators A*, vol. 105, pp. 211–224, 2003.

3. M.M. Yovanovich, J.R. Culham, and P. Teertstra, "Calculating Interface Resistance," *Electronics Cooling*, May, 1997 (http://electronics-cooling.com/articles/1997/may/article3.php)

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/microchannel_heatsink`

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/microchannel_heatsink_res`

Modeling Using the Graphical User Interface

The first part of this section describes how to build and solve the initial model, which does not account for the thermal contact resistance at the interface between the heat source and the heat sink.

MODEL NAVIGATOR

3 In the **Model Navigator** go to the **New** page. From the **Space dimension** list select **3D**.

4 In the list of application modes, select **Heat Transfer Module>Fluid-Thermal Interaction>Non-Isothermal Flow>Steady-state analysis**.

5 Click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu select **Constants**. Enter these names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T0	295[K]	Inlet air temperature
eta_air	18e-6[Pa*s]	Dynamic viscosity, air
k_air	27[mW/(m*K)]	Thermal conductivity, air
Cp_air	1006[J/(kg*K)]	Specific heat capacity, air
p0	1e5[Pa]	Outlet air pressure
k_ceramic	20.9[W/(m*K)]	Thermal conductivity, ceramic
Q_source	5[W/cm^3]	Heat production, ceramic

GEOMETRY MODELING

1 Go to the **Draw** menu and select **Work-Plane Settings**.

2 On the **Quick** page, select the **x-y** option button and in the **z** edit field type `2.85e-3`.

3 Click **OK**.

4 Shift-click the **Rectangle/Square** button on the Draw toolbar.

5 In the dialog box that appears, enter these properties; when done, click **OK**.

OBJECT DIMENSIONS	EXPRESSION
Width	`1e-3`
Height	`2e-4`
Base	Corner
x-position	0
y-position	0

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

7 In the same manner, create a second rectangle with these properties:

OBJECT DIMENSIONS	EXPRESSION
Width	`1e-3`
Height	`1e-4`
Base	Corner
x-position	0
y-position	0

8 From the **Draw** menu open the **Extrude** dialog box.

9 From the **Object selection** list select **R1**. In the **Distance** edit field type `-2.85e-3`, then click **OK**.

10 Click the **Geom2** tab.

11 Using the method of Steps 8 and 9, extrude the large rectangle, R1, once again but now by a **Distance** of `-1.85e-3`.

12 Click the **Geom2** tab.

13 Extrude the small rectangle, R2, by a **Distance** of `-1.65e-3`.

14 Click the **Geom2** tab.

15 Draw two new rectangles with the properties in the following tables:

OBJECT DIMENSIONS	EXPRESSION
Width	`5e-4`
Height	`1e-4`
Base	Corner
x-position	`2.5e-4`
y-position	0

OBJECT DIMENSIONS	EXPRESSION
Width	1e-3
Height	1e-4
Base	Corner
x-position	0
y-position	1e-4

16 Click the **Create Composite Object** button on the Draw toolbar.

17 In the **Object selection** list select **R3** and **R4**, the two rectangles you just created. Click **OK** to create their union, CO1.

18 From the **Draw** menu choose **Extrude**. Select the new composite object, CO1

19 In the **Distance** edit field type `-2.5e-4`, then click **OK**.

20 Click the **Create Composite Object** button.

21 In the **Set formula** edit field type `EXT1+EXT2+EXT3-EXT4`, then click **OK**.

22 Double-click the **AXIS** button on the status bar at the bottom of the user interface to hide the coordinate axes.

PHYSICS SETTINGS

1 Go to the **Options** menu and select **Expressions>Boundary Expressions**.

2 Select Boundary 10 and enter the following expression; when done, click **OK**.

NAME	EXPRESSION
`w_inlet`	`-0.3e16[1/(m^3*s)]*(2.5e-4+x)*(2.5e-4-x)*(1e-4+y)*(1e-4-y)`

This gives a parabolic inlet-velocity profile with a maximum inflow speed of 1.875 m/s and average inflow speed of 0.833 m/s.

Subdomain Settings—Weakly Compressible Navier-Stokes

1 From the **Multiphysics** menu select **Weakly Compressible Navier-Stokes (chns)**.

2 Go to the **Physics** menu and select **Subdomain Settings**.

3 Select Subdomains 1 and 2.

4 From the **Group** list, select **Solid domain**.

5 Select Subdomain 3 and enter `eta_air` in the η edit field.

6 Click the **Init** tab, then in the **p(t$_0$)** edit field type `p0`.

7 Click **OK**.

Boundary Conditions—Weakly Compressible Navier-Stokes

1 From the **Physics** menu select **Boundary Settings**.

2 Enter settings from the following table; when done, click **OK**.

SETTINGS	BOUNDARIES 7, 8, 23	BOUNDARIES 9, 11, 12, 16, 17, 18, 20	BOUNDARY 10	BOUNDARY 19
Boundary type	Symmetry boundary	Wall	Inlet	Outlet
Boundary condition		No slip	Velocity	Pressure, no viscous stress
u_0			0	
v_0			0	
w_0			`w_inlet`	
p_0				`p0`

Subdomain Settings—General Heat Transfer

1 Go to the **Multiphysics** menu and select **2 Geom1: General Heat Transfer (htgh)**.

2 In the **Physics** menu select **Subdomain Settings**.

3 Select Subdomain 1. From the **Group** list, select **Solid domain**.

4 Enter the following properties. When done, click **OK**.

PROPERTY	VALUE
k (isotropic)	`k_ceramic`
Q	`Q_source`

5 Select Subdomain 2. From the **Group** list, select **Solid domain**.

6 Click the **Load** button.

7 In the **Materials** list, select **Basic Material Properties>Aluminum**, then click **OK**.

8 Select Subdomain 3. On the **General** page, enter the following settings (for properties not listed, keep the default settings):

PROPERTY	VALUE
k (isotropic)	`k_air`
C_p	`Cp_air`

9 Click the **Convection** tab, then select **Ideal gas** from the **Matter state**list.

10 In the $\mathbf{p}_A$ edit field, type `p`.

11 Click the **Init** tab.

12 Select all three subdomains, and in the **Temperature** edit field enter `T0`.

13 Click **OK**.

Boundary Conditions—General Heat Transfer

1. Go to the **Physics** menu and select **Boundary Settings**.
2. Enter the settings from the following table; when done, click **OK**.

SETTINGS	BOUNDARY 10	BOUNDARY 19
Boundary condition	Temperature	Convective flux
T_0	T0	

For all other boundaries, keep the default setting, Insulation/Symmetry.

MESH GENERATION

1. From the **Mesh** menu select **Free Mesh Parameters**.
2. Click the **Boundary** tab and select Boundaries 10, 12, and 16–20.
3. In the **Maximum element size** edit field type `9e-5`.
4. Go the **Advanced** page, then in the **y-direction scale factor** edit field type `5`.
5. Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

By default, COMSOL Multiphysics solves 3D models with Navier-Stokes as the ruling application mode using the GMRES iterative solver. However, for models (such as this one), with less than 100,000 degrees of freedom, the direct PARDISO solver is more efficient. Therefore, change the default settings according to the following instructions.

1. Click the **Solver Parameters** button on the Main toolbar.
2. On the **General** page of the **Solver Parameters** dialog box find the **Linear system solver** list and select **Direct (PARDISO)**.
3. Click **OK** to close the **Solver Parameters** dialog box.
4. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the temperature field. To create Figure 2-44, which also shows the heat-flux streamlines, follow these steps:

1. Click the **Plot Parameters** button on the Main toolbar.

2 Go to the **Streamline** page and select the **Streamline plot** check box.

3 In the **Predefined quantities** list select **General Heat Transfer (htgh)>Total heat flux**.

4 In the **Streamline plot type** list select **Magnitude controlled**.

5 On the **Density** page set the **Min distance** to `0.02` and the **Max distance** to `0.12`.

6 Click the **Line Color** tab. Select the **Uniform color** option button, then click the **Color** button to launch the **Streamline Color** dialog box. On the **Swatches** page, select a yellow color, then click **OK**.

7 Back on the **Streamline** page, select **Tube** from the **Line type** list, then click the **Tube Radius** button.

8 In the **Tube Radius Parameters** dialog box, clear the **Auto** check box for the **Radius scale factor**, then type `0.3` in the corresponding edit field. Click **OK**.

9 Go to the **Slice**.

10 In the **Predefined quantities** list select **General Heat Transfer (htgh)>Temperature**.

11 Click **Apply**.

The following steps describe how to create Figure 2-45, which shows the velocity field and the velocity streamlines:

1 Return to the **Plot Parameters** dialog box.

2 On the **Streamline** page, in the **Predefined quantities** list select **Weakly Compressible Navier-Stokes (chns)>Velocity field**.

3 Go to the **Slice** page and in the **Predefined quantities** list select **Weakly Compressible Navier-Stokes (chns)>Velocity field**.

4 Click **OK**.

Modeling Using the Graphical User Interface—Extended Model

To build the extended model, which accounts for the thermal contact resistance, continue with these steps:

OPTIONS AND SETTINGS

1 From the **Options** menu select **Constants**. Add this constant; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
h_j	5400[W/(m^2*K)]	Thermal joint conductivity

GEOMETRY MODELING

1 Click the **Draw Mode** button on the Main toolbar.
2 Select the geometry and click the **Split Object** button in the Draw toolbar.
3 Select the geometry objects CO3 and CO4, then click the **Union** button on the Draw toolbar.
4 Select the geometry objects CO1 and CO2, then click the **Create Pairs** button on the Draw toolbar.

Boundary Conditions—General Heat Transfer (htgh)

1 Go to the **Physics** menu and select **Boundary Settings**.
2 Click the **Pairs** tab and select Pair 1 (identity).
3 Select **Thin thermally resistive layer** from the **Boundary condition** list.
4 Enter the settings from thus table; when done, click **OK**.

SETTINGS	VALUE
k_{res}	`h_j*1.0[m]`
d_{res}	`1.0[m]`

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the temperature field and streamlines for the total heat flux. To create Figure 2-46, remove the streamlines with the following steps:

1 Click the **Plot Parameters** button on the Main toolbar.
2 On the **Streamline** page, clear the **Streamline plot** check box.
3 Click **OK**.

Heat Transfer in a Surface-Mount Package for a Silicon Chip

Introduction

All integrated circuits—especially high-speed devices—produce heat. In today's dense electronic system layouts heat sources are many times placed close to heat-sensitive ICs. Designers of printed-circuit boards often need to consider the relative placement of heat-sensitive and heat-producing devices, so that the sensitive ones do not overheat.

One type of heat-generating device is a voltage regulator, which can produce several watts of heat and reach a temperature higher than 70 °C. If the board design places such a device close to a surface-mounted package that contains a sensitive silicon chip, the regulator's heat could cause reliability problems and failure due to overheating.

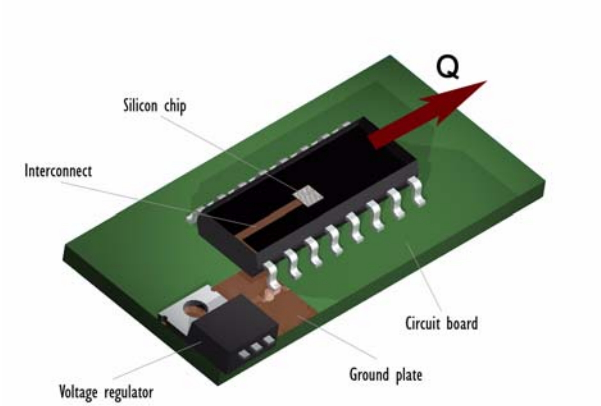


Figure 2-47: Layout of the simulated silicon device, its package, and a voltage regulator. The chip and the voltage regulator are connected through a ground plane, a pin, and the interconnect.

This simulation investigates the thermal situation for a silicon chip in a surface-mount package placed on a circuit board close to a hot voltage regulator. The chip is subjected to heat from the regulator and from internally generated heat.

Model Definition

The model is based on a SMD IC and voltage regulator layout as in Figure 2-47. The silicon chip sits in the center of the package and dissipates its heat to the surrounding environments. The chip also connects to a ground plane through an interconnect and one of the pins. A heat generating voltage regulator is placed on the same ground plane. This means that the voltage regulator may affect the silicon chip by the conducted heat and this may lead to overheating of the chip.

Heat transfers through the mounted package to the surroundings through conduction according to:

$$\nabla \cdot (-k \nabla T) = Q \, .$$

Q is negligible in the circuit board, pins and package, while in the chip this model sets that parameter to a value equivalent to 20 mW. The conductivities of the components are chosen to be similar to:

- silicon, for the chip
- aluminum, for the pins
- FR4, for the pc board
- copper, for the ground plane and interconnect
- an arbitrary plastic, for the chip package

Heat dissipates from all air-exposed surfaces through forced heat convection, which is modeled using a heat transfer coefficient, h:

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\mathrm{inf}} - T)$$

The voltage regulator is simulated by setting a fixed temperature at that surface. The thin conducting layers of the ground plane and interconnect within the package is modeled using a 2D shell approximation, according to:

$$\nabla \cdot (-d_{\mathrm{s}} k \nabla_t T) = 0$$

where d_{s} is the layer's thickness, and ∇_t represents the nabla operator projected onto the direction of the plane. The model uses a General Heat Transfer application mode to describe the 3D heat transfer as well as the 2D shell heat transfer.

Results and Discussions

Figure 2-48 illustrates the temperature distribution through the thickness. Being a good conductor, the interconnect delivers heat to the outer edge of the package, which gives the fairly constant temperature distribution around the interconnect.

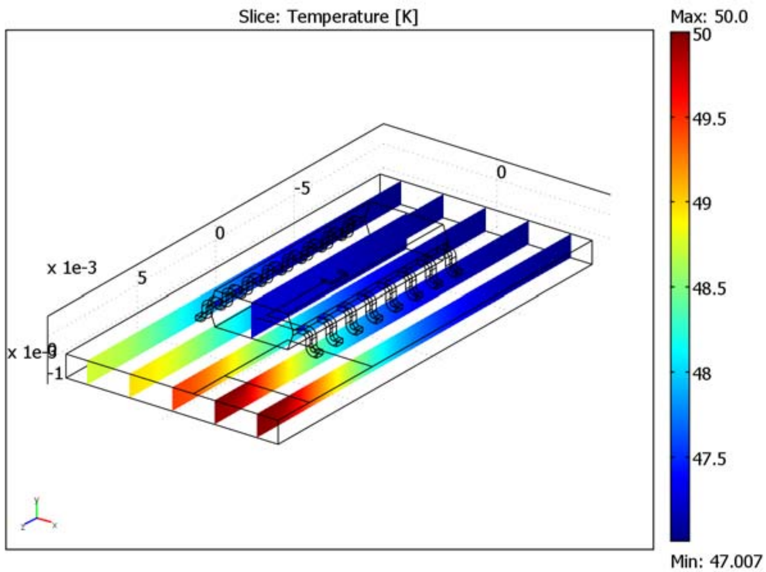


Figure 2-48: Slice plot of the temperature through the circuit board, interconnect, chip, and package. The effect of the interconnect is evident by its ability to conduct heat from the chip to the outer parts of the package.

An alternative view is achieved by using the transparency feature in the postprocessing tools of COMSOL Multiphysics. This results in a transparent 3D view of the temperature distribution, as depicted in Figure 2-49. In that figure you can see the temperature distribution around the chip and along the interconnect.

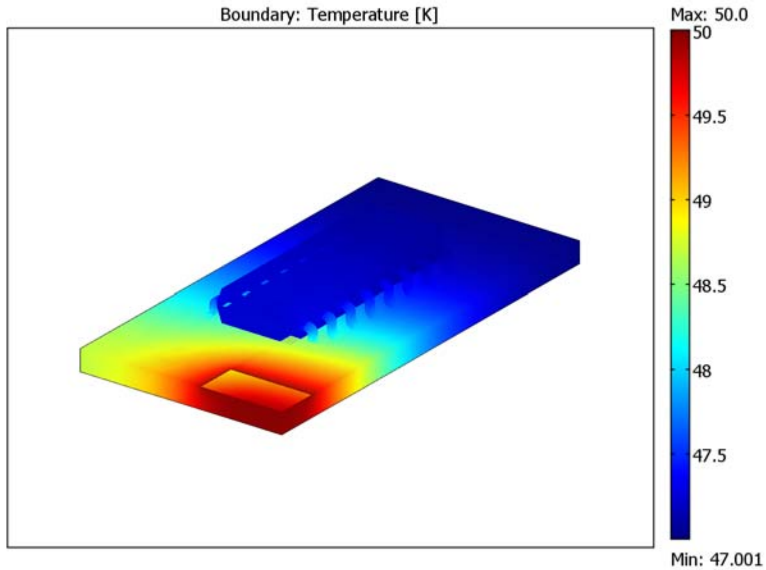


Figure 2-49: Boundary plot of the temperature created with the assistance of the transparency tool in COMSOL Multiphysics. This view also gives the temperature distribution on the chip and along the interconnect.

To get a closer look at the stationary temperature of the silicon chip, plot the temperature at the bottom boundary of the chip.

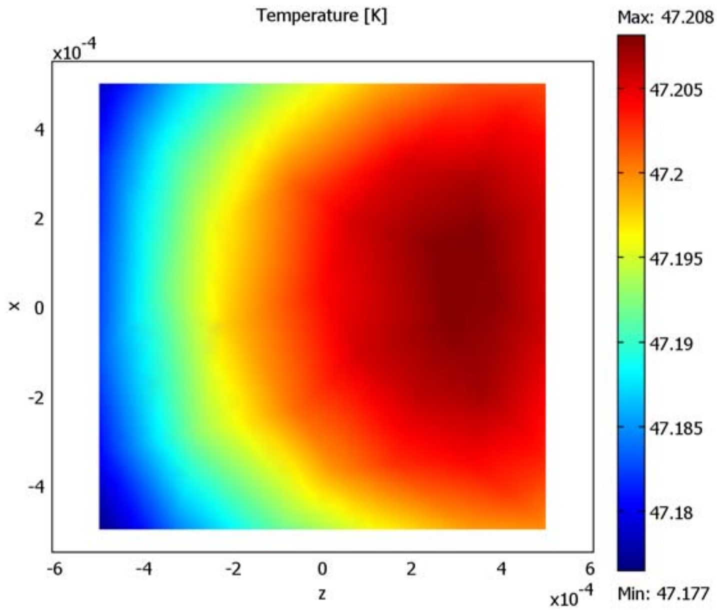


Figure 2-50: Temperature distribution on the bottom surface of the silicon chip.

The simulation predicts a maximum temperature of the silicon device of 46.6 °C. This means that the device will not overheat in the present configuration.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/surf_mount_pack`

Modeling in COMSOL Multiphysics

This model uses the General Heat Transfer application mode from the Heat Transfer Module, and that application mode allows the definition of highly conductive layers.

They are thin layers that conduct heat well so you need not define them in 3D. The two layers that have this definition are:

- The interconnect between the chip and the grounded pin.
- The ground plane that is also thermally connected to the temperature constraint coming from the voltage regulator.

While the numerical method considers these two modeling domains as interior boundaries, the model still includes a thickness to take the 3D heat flux into account.

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

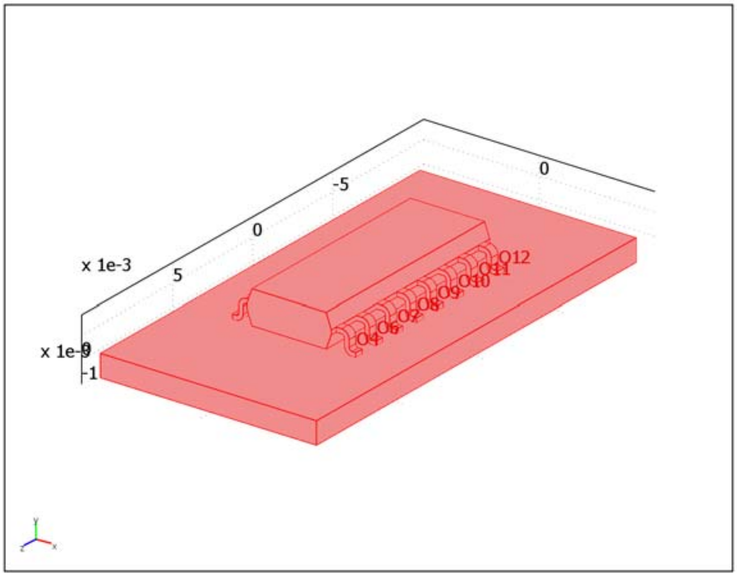
1 Start COMSOL Multiphysics and get to the **Model Navigator** by double-clicking the COMSOL Multiphysics icon on the Windows desktop or, on Unix/Linux systems, enter the command `comsol`.
2 Click the **New** tab, and in the **Space dimension** list select **3D**.
3 This problem is ideally suited for the Heat Transfer Module. Therefore, go to the list of application modes and select **Heat Transfer Module>General Heat Transfer>Steady-state analysis**, and click **OK**.

CREATING THE GEOMETRY

The 3D workspace is now ready. You can, of course, create the 3D geometry with the built-in CAD tools of COMSOL Multiphysics, but an interesting alternative is to import a ready-made geometry from a dedicated CAD tool.

1 From the **File** menu select **Import>CAD Data From File.** Find the file `surf_mount_pack.mphtxt` or `surf_mount_pack.igs` (if you have the CAD Import Module) and then click **OK**.

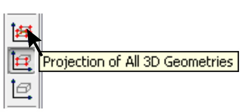
2 To get a good view, rotate the geometry with the mouse and then click the **Zoom Extents** button on the Main toolbar. The geometry in the drawing area on your screen should look like that in the figure below.

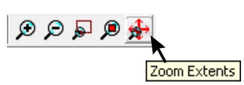


To this base geometry you must add the interconnect between the pin and the chip as well as the ground plane and the temperature surface resulting from the voltage regulator. These details are all 2D surfaces that you can best add on work planes. Thus, start by adding the work plane for the interconnect, which is on the zx-plane at $y = 0$.

3 Choose **Draw>Work-Plane Settings**, and on the **Quick** tab choose **z-x** and click **OK**.

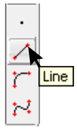
4 To see the 3D geometry projected on the 2D plane, click the **Projection of All 3D Geometries** on the Draw toolbar, and then click the **Zoom Extents** on the Main toolbar.





5 To simplify the drawing of the 2D object, and add some extra grid points. Choose **Options>Axes/Grid Settings**, then click the **Grid** tab. Clear the **Auto** check box. In the following edit fields enter the these values: **x-spacing:** `0.002`, **Extra x:** `5e-4 4.245e-3 4.645e-3`, **y-spacing:** `0.002`, and **Extra y:** `-2e-4 2e-4`. Click **OK**.

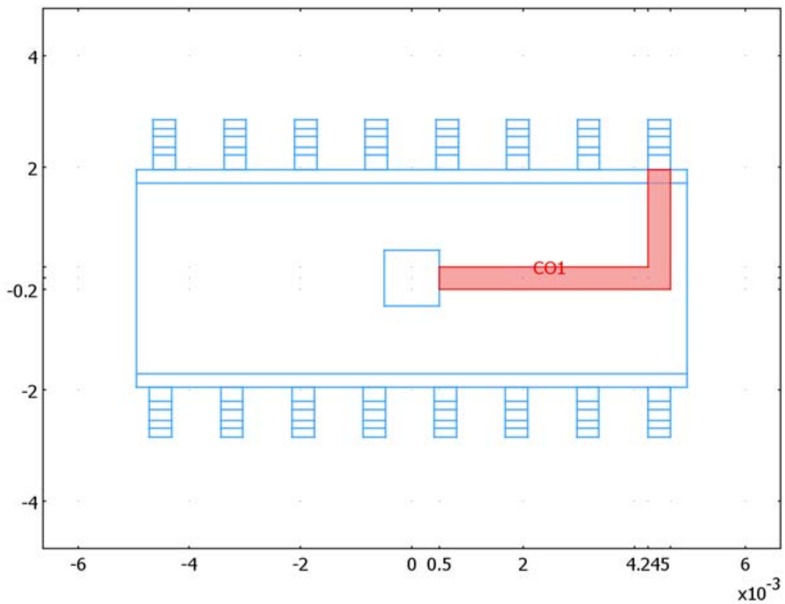
6 Press Shift and click the **Line** button on the Draw toolbar.



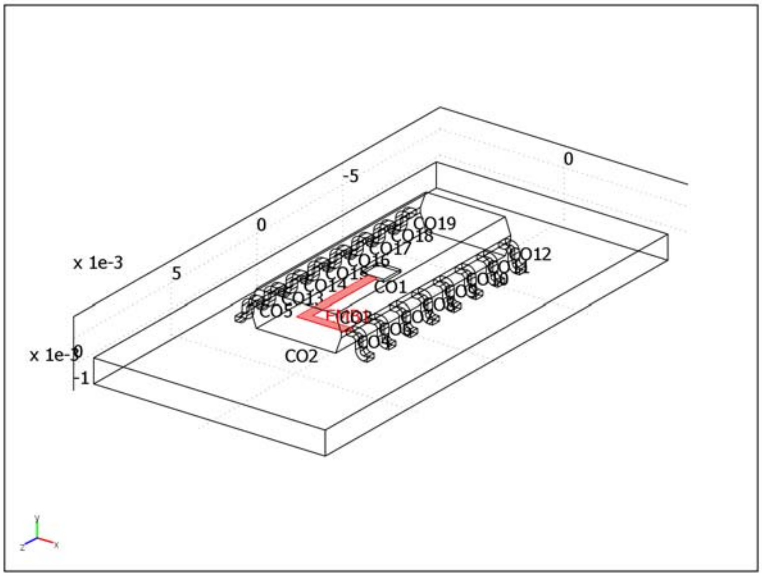
7 In the **Line** dialog box, go to the **Coordinates** area and type in the following data:

x	5e-4 5e-4 4.645e-3 4.645e-3 4.245e-3 4.245e-3 5e-4
y	2e-4 -2e-4 -2e-4 1.95e-3 1.95e-3 2e-4 2e-4

8 From the **Style** list, select **Closed polyline (solid)** and then click **OK** to generate the interconnect between the chip and the connector as depicted in the following figure.



9 To embed the object CO1 in the 3D workspace, choose **Draw>Embed**, then click **OK**.



Now, finalize the geometry by adding the ground plane for the voltage regulator.

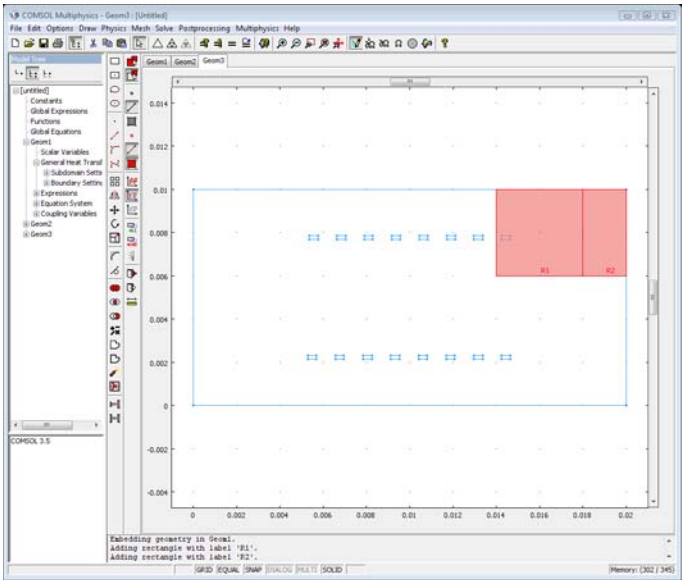
10 Choose **Draw>Work-Plane Settings**.

11 In the **Work-Plane Settings** dialog box, click the **Add** button.

You do not know for certain the exact position of the circuit board's top surface, so project this surface to the work plane and then use the fixed positions of this projection's vertices to draw the two surfaces.

12 Click the **Vertices** tab. Select three vertices on the corners of the circuit board's top surface by clicking at one point at a time and adding it to the list using the **>>** button. Add the points in this order: `CO2:3`, `CO2:4`, `CO2:8`. This gives the work plane the proper orientation for the next step. Click **Apply** to view the work plane axis. Click **OK**, then click the **Zoom Extents** button on the Main toolbar.

13 Click the **Rectangle/Square** button on the Draw toolbar to draw the two rectangles shown in the next figure. Use the existing grid points to snap the rectangles while drawing them. Press Ctrl+A to select both of them.



14 Select the menu item **Draw>Embed** and then click **OK**.

PHYSICS SETTINGS

Subdomain Settings

This section deals with the setting the material's physical properties.

1 Choose **Physics>Subdomain Settings** and press Ctrl+A to select all subdomains.

2 Click the **Element** tab and select **Lagrange - Linear** from the **Predefined elements** list. Doing so saves computation time and memory.

3 Click the **General** tab and then the **Load** tab to load a predefined material property set. Select **Aluminum** in the list, then click **OK**.

   In COMSOL Multiphysics you can enter individual numerical values of material properties such as thermal conductivity directly in the **Subdomain Settings** dialog box. An alternative is to use predefined materials from the materials library. The next step shows how to add some new materials to the materials library.

4 Choose **Options>Materials/Coefficients Library**. Click **New** and in the **Name** field type `PCB (FR4)`. Also set the thermal conductivity **k** to `0.3` (W/(m·K)). Click **OK**. Complete the new material additions as in the following list.

MATERIAL NAME	THERMAL CONDUCTIVITY, K
`PCB (FR4)`	`0.3`
`Plastic`	`0.2`

5 Return to the **Subdomain Settings** dialog box and highlight Subdomain 1 (the circuit board), then go to the **Library material** list and select **PCB (FR4)**. Next select Subdomain 10 (the package) and select **Plastic** from the materials menu. Finally select Subdomain 11 (the chip) and click the **Load** button to load the **Silicon** material property set from the built-in list, and click **OK**.

6 Keep the chip highlighted and add an internal heat source (W/m^3) by going to the **Q** edit field and entering `2e8`, which corresponds to 20 mW for the whole volume of the device. Click **OK**.

Boundary Conditions

1 Select the menu item **Physics>Boundary Settings** and press Ctrl+A to select all boundaries to the exterior (the interior boundaries are grayed out).

2 To set a cooling rate by assuming a heat transfer coefficient of 50 W/(m^2·K) to the surroundings at a temperature of 30 °C, set the **Boundary condition** to **Heat flux** and then set **h** to `50` and $\mathbf{T_{inf}}$ to `30[degC]`.

3 Select the surface to which the voltage regulator is connected (Boundary 141), then select **Temperature** from the **Boundary condition** list. In the $\mathbf{T_0}$ edit field, type `50[degC]`.

4 Select the ground plane (Boundary 140), then click the **Highly Conductive Layer** tab. Select the check box **Enable heat transfer in highly conductive layer**. Specify the layer thickness $\mathbf{d_s}$ as `1e-4`. The other material properties are those of copper, which is what the ground plane is made of.

5 Select the **Interior boundaries** check box and select the interconnect (Boundary 138). Again select the check box **Enable heat transfer in highly conductive layer** and here specify a layer thickness $\mathbf{d_s}$ of `5e-6`. Click **OK**.

GENERATING THE MESH

1 Choose **Mesh>Free Mesh Parameters**.

2 Click the **Boundary** tab and select the surfaces for the ground plane and voltage regulator (Boundaries 140 and 141). On each of these boundaries set the **Maximum element size** to `1e-4`. Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

Because this model has a limited size you can use a direct solver. Also, for two reasons, you can utilize that the system matrices become symmetric: The model only involves conduction in the subdomain, and the boundary conditions only depend on the temperature variable T.

1 Choose **Solve>Solver Parameters**.

2 Select **Direct Cholesky (TAUCS)** from the **Linear system solver** list, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

VISUALIZING THE SOLUTION

The default plot shows a slice plot of the temperature throughout a number of cross sections in the device. Note that the temperature is displayed in Kelvin, which is the default temperature unit in the SI system.

1 To generate Figure 2-49, click the **Plot Parameters** button on the top toolbar. On the **General** page, clear the **Slice** check box and select the **Boundary** check box.

2 Go to the **Boundary** page and select **°C** from the **Unit** list.

3 Click **OK** to display the boundary plot.

4 To see the interior, click the **Increase Transparency** button on the lower part of the left toolbar.

5 To generate Figure 2-50, first choose **Postprocessing>Domain Plot Parameters**.

6 Click the **Surface** tab and select Boundary 134 (bottom plane of the chip) from the list.

7 Select **°C** from the **Unit** list to display the result in degrees Celsius.

8 Select **zx-plane** from the list of **x- and y-axis data**, then click **OK**.

Rapid Thermal Annealing

Introduction

In the semiconductor industry, rapid thermal annealing (RTA) is a semi-conductor process step used for the activation of dopants and the interfacial reaction of metal contacts. In principle, the operation involves rapid heating of a wafer from ambient to approximately 1000–1500 K. As soon as the wafer reaches this temperature, it is held there for a few seconds and then finally quenched. A rapid process step is crucial in order to avoid too much diffusion of the dopants. Furthermore, it is also important to avoid overheating and nonuniform temperature distribution to occur. An RTA apparatus uses high-power IR lamps as heat sources (Ref. 1).

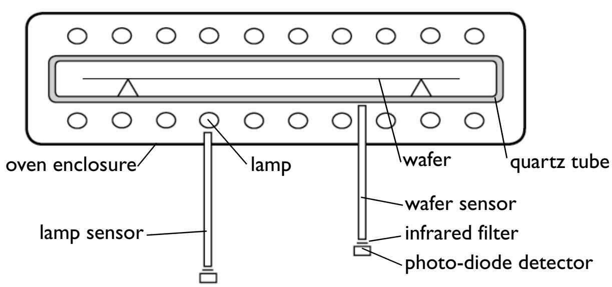


Figure 2-51: Diagram of a typical RTA (rapid thermal annealing) apparatus.

A technical difficulty lies in how to properly measure the wafer's temperature during the process. Two commonly used technical solutions are: thermocouples and IR sensors.

To achieve an accurate measurement, it is important that the temperature sensor not be subjected to direct radiation from the lamp. Ideally positioned, the sensor only receives secondary radiation; that is, the radiation reflected and emitted by the silicon wafer. Desirable characteristics of the sensor are high accuracy and short response time. While a high-performance design requires superior electronics, the sensor geometry plays a big role. In a nutshell, the sensor needs to be large enough to register a sufficient amount of radiation but light enough to minimize its own thermal inertia. Since COMSOL Multiphysics gives you control over the geometry, a parameter optimization of the sensor could be an exciting project. But first, justify that an infrared sensor is indeed more appropriate than the inexpensive thermocouple.

Model Definition

Figure 2-51 illustrates a typical RTA configuration. In many applications, RTA makes use of double-sided heating, in which IR lamps are positioned both above and below the silicon wafer. In this example we are modeling a single-sided heating apparatus, as depicted in Figure 2-52.

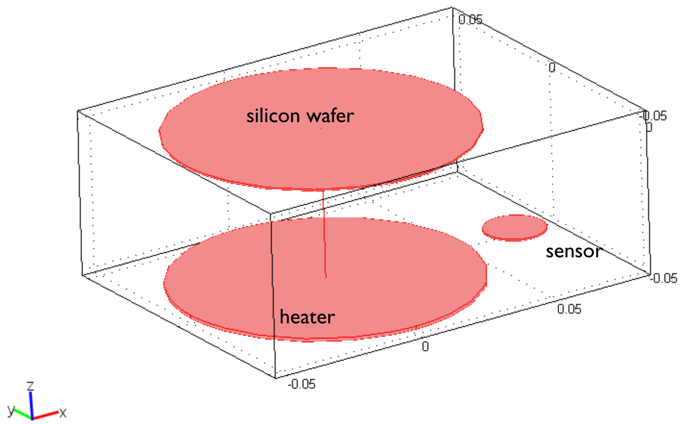


Figure 2-52: The model geometry.

The components in Figure 2-52 are contained in a chamber with temperature-controlled walls with a set point of 400 K. This results in a closed cavity so you can omit the geometry of the chamber walls. Furthermore, the model assumes that this physical system is dominated by radiation and convection cooling. The convective cooling of the wafer and sensor to the gas (at 400 K) is modeled using a heat transfer coefficient, h (in this example set to 20 W/(m^2·K)).

The problem is governed by the heat equation, given below together with its boundary conditions:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = Q$$

$$-\mathbf{n} \cdot (-k \nabla T) = h(T_{\mathrm{inf}} - T) + (\varepsilon / (1 - \varepsilon))(J_0 - \sigma T^4)$$

Here ρ is the density; k denotes the thermal conductivity; Q represents the volume heat source; $\mathbf{n}$ is the surface normal vector; T_{inf} equals the temperature of the convection cooling gas; ε denotes the surface emissivity; J_0 is the expression for surface radiosity (further described in the *Heat Transfer Module User's Guide*); and σ is the Stefan-Boltzmann constant.

The model simulates the lamp as a solid object with a volume heat source of 25 kW. It is insulated on all surfaces except the for the top, which faces the silicon wafer. At this surface, heat leaves the lamp as radiation only. In order to capture the lamp's transient startup time, the model uses a low heat capacity, C_p, for the solid (10 J/(kg·K)). The lamp's other thermal properties are identical to those of copper metal (the default value in the application mode).

In this case assume that the wafer dissipates energy via radiation and convection on all surfaces. The sensor is insulated at all surfaces except the top, which is subjected to both convection and radiation. The thermal material properties are set to those of alumina.

The following table summarizes the material properties used in the model:

TABLE 2-6: MATERIAL PROPERTIES

MATERIAL	k (W/(m·K))	ρ (kg/m^3)	C_p (J/(kg·K))	ε
IR lamp	400	8700	10	0.99
Silicon wafer (silicon)	163	2330	703	0.5
Sensor	27	2000	500	0.8

The model simulates the transient temperature field for 10 s of heating. The initial temperature is 400 K for all objects.

Results and Discussion

Figure 2-53 displays the temperature distribution after 10 s of heating.

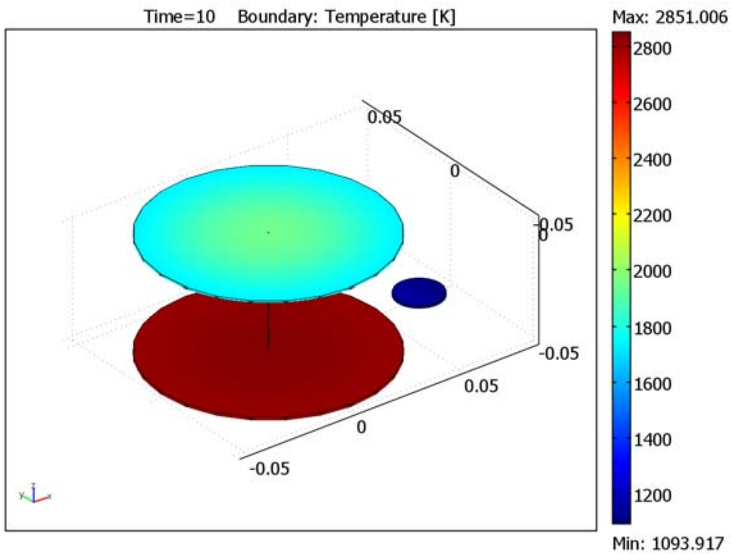


Figure 2-53: Temperatures of the lamp, wafer, and sensor after 10 s of heating.

After 10 seconds, the temperatures of the wafer and sensor differ significantly: the wafer is at 1800 K, whereas the sensor is at 1100 K.

Notice that the temperature distribution in the wafer with a delta of several hundred degrees is not very uniform, and that you probably can do much better by reconfiguring the heat source. However, such a reconfiguration is not included in this model.

To investigate how well the sensor's temperature reflects that of the wafer surface, it is useful to plot the temperature transient of the wafer surface's centerpoint that faces the lamp (T_{wafer}), together with the temperature at a point on the sensor top surface (T_{sensor}) (see Figure 2-54).

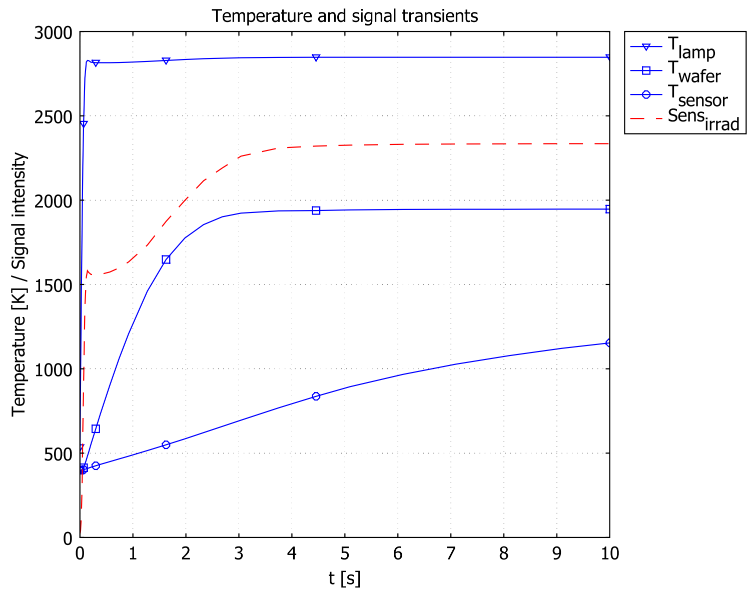


Figure 2-54: The temperature transients of the lamp, the silicon wafer, and the sensor, together with the irradiation power at the sensor surface.

The sensor temperature reflects that of the silicon wafer poorly. This means that the signal of a thermocouple, positioned anywhere in the sensor domain of Figure 2-52, is of little use for regulating this process.

The IR-detector transient (Sens_{irrad}) matches the wafer temperature characteristic quite well. A scalar amplification allows for a high accuracy measurement of the wafer temperature. The precise amplification factor is system-dependent and subject to a calibration requirement.

However, IR-sensor methodology also has drawbacks. The IR signal depends on the emissivity of the wafer, which will vary with temperature making the response nonlinear. Furthermore, the IR signal is very sensitive to geometry changes.

The bright side is that COMSOL Multiphysics does not set any limits with respect to these phenomena and allows you to study them fully.

Reference

1. A.T. Fiory, "Methods in Rapid Thermal Annealing," Proc. 8th Int'l Conf. Advanced Thermal Processing of Semiconductors (RTP 2000), http://web.njit.edu/~fiory/Papers/RapidThermalAnnealing00.pdf, pp. 15–25.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/thermal_anneal`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**.

2 From the **Space dimension** list, select **3D**.

3 In the list of application modes select **Heat Transfer Module>General Heat Transfer>Transient analysis**, then click **OK**.

CONSTANTS AND EXPRESSIONS

1 From the **Options** menu, select **Constants**.

2 Define the following constants; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_wall	400[K]	Temperature, wall
T_gas	400[K]	Temperature, gas
h_gas	20[W/(m^2*K)]	Heat transfer coefficient
k_sens	27[W/(m*K)]	Thermal conductivity, sensor
rho_sens	2000[kg/m^3]	Density, sensor
Cp_sens	500[J/(kg*K)]	Heat capacity, sensor
e_sens	0.8	Surface emissivity, sensor
e_lamp	0.99	Surface emissivity, lamp
q_lamp	25[kW]/(pi*50^2*1[mm^3])	Heating power, lamp
e_wafer	0.5	Surface emissivity, wafer
Cp_lamp	10[J/(kg*K)]	Heat capacity, lamp
ampl	50	Amplification factor

GEOMETRY MODELING

1 Create three cylinders. To do so, open the menu item **Draw>Cylinder** and enter these settings; when finished, click **OK**.

OBJECT	RADIUS	HEIGHT	AXIS BASE POINT, X	AXIS BASE POINT, Z
CYL1	0.05	5e-4	0	0
CYL2	0.05	1e-3	0	-5e-2
CYL3	1e-2	1e-3	0.07	-5e-2

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

PHYSICS SETTINGS

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 In the **General** page, select Subdomain 1. In the **Cp** edit field type `Cp_lamp`, and in the **Q** edit field type `q_lamp`. Use the default values for both the conductivity and the density.

3 Select Subdomain 2, then click the **Load** button. In the **Material** list, select **Basic Material Properties>Silicon**. Click **OK**.

4 Select Subdomain 3. In the **k** edit field type `k_sens`, in the ρ edit field type `rho_sens`, and in the $\mathbf{C_p}$ edit field type `Cp_sens`.

5 Select all subdomains. Click the **Init** tab, then in the $\mathbf{T(t_0)}$ edit field type `T_wall`.

6 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select Boundary 4. In the **Boundary condition** list select **Heat flux**, and in the **Radiation type** list select **Surface-to-surface**. In the ε edit field type `e_lamp`, and in the $\mathbf{T_{amb}}$ edit field type `T_wall`.

3 Select Boundaries 5–8, 10, 12, and 16.

4 In the **Boundary condition** list select **Heat flux**. In the **h** edit field type `h_gas`, and in the $\mathbf{T_{inf}}$ edit field type `T_gas`.

5 In the **Radiation type** list select **Surface-to-ambient**. In the ε edit field type `e_wafer`, and in the $\mathbf{T_{amb}}$ edit field type `T_wall`.

6 Select Boundaries 7 and 16. From the **Radiation type** list, select **Surface-to-surface**.

7 Select Boundary 16 alone. Change the entry in the ε edit field to `e_sens`. Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.
2 In the **Predefined mesh sizes** list, select **Coarser**. Click the **Advanced** tab. In the **z-direction scale factor** edit field type `5`. Click the **Remesh** button, then click **OK**.

PREPARE POSTPROCESSING

To prepare some postprocessing operations, you need to define an integration coupling variable. In addition, a line intersecting the lamp and wafer surfaces is also helpful in later postprocessing.

1 Choose **Options>Integration Coupling Variables>Boundary Variables**.
2 Select Boundary 16. In the **Name** edit field type `Sens_irrad`, and in the **Expression** edit field type `G_htgh` (a predefined application mode variable representing inward radiation which includes both surface-to-surface and surface-to-ambient contributions). Click **OK**.
3 From the **Draw** menu select **Line.** In the edit fields for **x**, **y**, and **z** type `0 0`, `0 0`, and `-5e-2 1e-3`, respectively. Click **OK**. (This step is not necessary if you loaded the geometry file).

COMPUTING THE SOLUTION

1 From the **Solve** menu, select **Solver Parameters**.
2 On the **General** page, type `0 10` in the **Times** edit field.
3 In the **Linear system solver** list, select **Direct (UMFPACK)**.
4 Click the **Time Stepping** tab. In the **Times to store in output** list, select **Time steps from solver**.
5 From the **Consistent initialization of DAE systems** list, select **On**.
6 From the **Error estimation strategy** list, select **Exclude algebraic**. Click **OK**.

   The last setting instructs the solver to omit the radiation calculations, which is always a stationary solution (algebraic equation), from the time-stepping error analysis. This greatly speeds up the solution process in terms of time stepping.
7 Click the **Solve** button on the Main toolbar (the solving process should take less than a minute).

POSTPROCESSING AND VISUALIZATION

Generate Figure 2-53 by executing the following instructions:

1 From the **Postprocessing** menu open the **Plot Parameters** dialog box.

2 On the **General** page, clear the **Slice** check box and select the **Boundary** check box in the **Plot type** area. Click **OK**.

The following steps generate Figure 2-54:

1 From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box.

2 On the **General** page, select the **Keep current plot** check box.

3 On the **Point** page, select Point 10.

4 In the **y-axis data** area, verify that the selection in the **Predefined quantities** list is **Temperature**.

5 Click the **Line Settings** button. In the **Line marker** list, select **Triangle**.

6 Select the **Legend** check box, then click **OK**.

7 Click **Apply** to plot the lamp temperature.

8 Select Point 12, then click the **Line Settings** button.

9 In the **Line marker** list select **Square**, then click **OK**.

10 Click **Apply** to add the plot of the wafer temperature.

11 Select Point 23, then click the **Line Settings** button.

12 In the **Line marker** list select **Circle**, then click **OK**.

13 Click **Apply** to generate the sensor temperature plot.

Finally, add the IR-detector transient to the plot:

14 In the **Expression** edit field, type `Sens_irrad*ampl`.

15 Click the **Line Settings** button.

16 Set the **Line color** to **Color**, the **Line style** to **Dashed line**, and the **Line marker** to **None**. Click **OK**.

17 Click **OK** to generate the plot and close the **Domain Plot Parameters** dialog box.

18 Click the **Edit Plot** toolbar button in the figure window.

19 Select the first **Line** object in the **Axes** tree on the left, then type `T<sub>lamp</sub>` in the **Legends** dialog box.

20 Repeat the previous step for the remaining three line objects, entering the legends `T<sub>wafer</sub>`, `T<sub>sensor</sub>`, and `Sens<sub>irrad</sub>`, respectively.

21 Click **OK** to close the **Edit Plot** dialog box and finish the plot.

Thermo-Photo-Voltaic Cell

Introduction

The following example illustrates an application that maximizes surface-to-surface radiative fluxes and minimizes conductive heat fluxes.

A thermo-photo-voltaic (TPV) cell generates electricity from the combustion of fuel and through radiation (Ref. 1). Figure 2-55 depicts the general operating principle. The fuel burns inside an emitting device that radiates intensely. Photo-voltaic (PV) cells—almost like solar cells—capture the radiation and convert it to electricity. The efficiency of a TPV device ranges from 1% to 20%. In some cases, TPVs are used in heat generators to co-generate electricity, and the efficiency is not so critical. In other cases TPVs are used as electric power sources, for example in automobiles (Ref. 2). In those cases efficiency is a major concern.

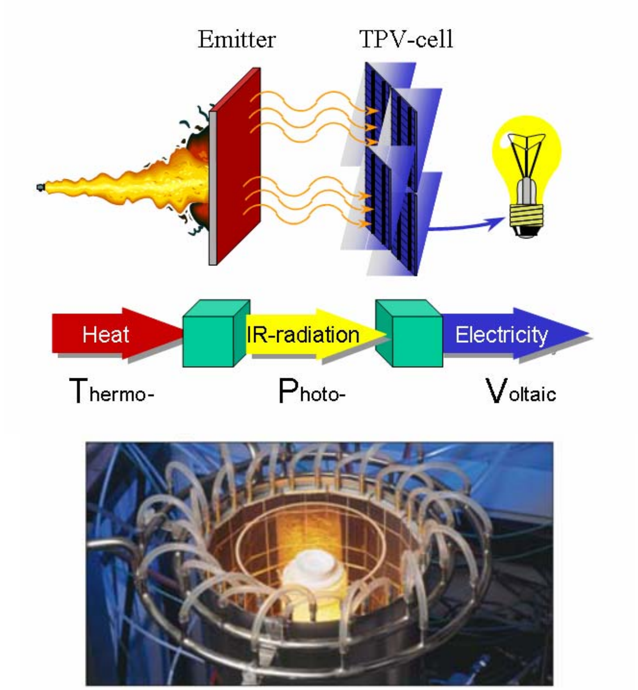


Figure 2-55: Operating principle of a TPV device (Ref. 3), and an image of a prototype system (Ref. 4).

TPV systems, unlike typical electronic systems, must maximize radiation heat transfer to improve efficiency. However, inherent radiation losses—radiation not converted to electric power—contributes to the PV cells' increased temperature. Further, heat transfer through conduction results in increased cell temperature. PV cells have a limited operating temperature range that depends on the type of material used. Solar cells are limited to temperatures below 80 °C, whereas high-efficiency semiconductor materials can withstand as much as 1000 °C. Photovoltaic efficiency is often a function of temperature with a maximum at some temperature above ambient.

To improve system efficiency, engineers prefer to use high-efficiency PV cells, which however can be quite expensive. To reduce system costs, engineers work with smaller-area PV cells and then use mirrors to focus the radiation on them. However, there is a limit for how much you can focus the beams; if the radiation intensity becomes too high, the cells can overheat. Thus engineers must optimize system geometry and operating conditions to achieve maximum performance at minimum material costs.

The following model, which uses the General Heat Transfer application mode, investigates the influence of operating conditions (flame temperature) on system efficiency and the temperature of components in a typical TPV system. The model can also assess the influence of geometry changes.

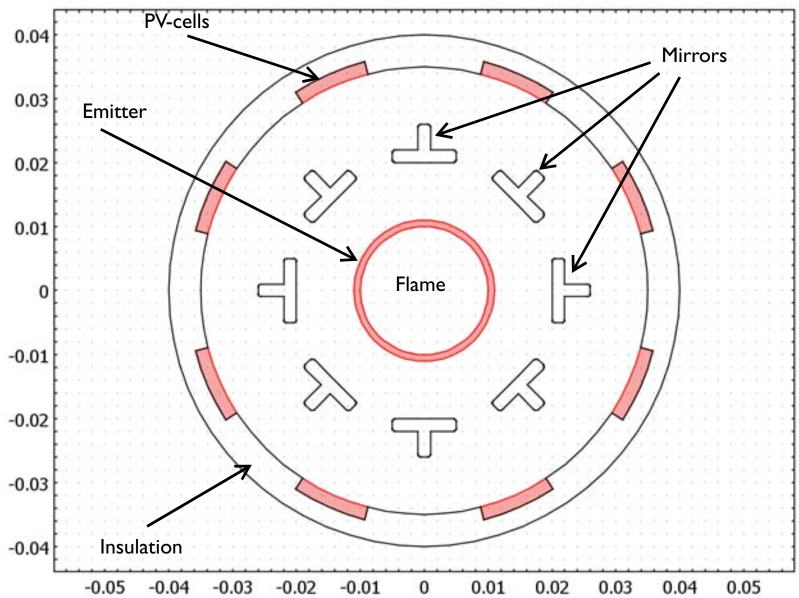


Figure 2-56: Geometry and dimensions of the modeled TPV system.

Figure 2-56 depicts the geometry and dimensions of the system under study. To reduce the temperature, the PV cells are water cooled on their back side (at the interface with the insulation).

The following equation describes the heat fluxes, radiative flux, and conductive flux; after it comes the boundary condition equation

$$\rho C_p \frac{\partial T}{\partial t} + \nabla(-k\nabla T) = Q$$

$$-\mathbf{n} \cdot (-k\nabla T) = h(T_{inf} - T) + (\varepsilon/(1-\varepsilon))(J_0 - \sigma T^4) + q$$

where ρ is the density, k denotes the thermal conductivity (W/(m·K)), Q represents the volume heat source (W/m^3), $\mathbf{n}$ is the surface normal vector, h is the convective heat transfer film coefficient (W/(m^2·K)), T_{inf} equals the temperature of the convection coolant, ε equals the surface emissivity, J_0 is the surface radiosity expression (W/m^2, further described in the *Heat Transfer Module User's Guide*), and σ equals the Stefan-Boltzmann constant.

Conduction is always present on the different boundaries. The model simulates the emitter with a specific temperature, T_{heater}, on the inner boundary. At the outer emitter boundary, it takes radiation (surface-to-surface) into account in the boundary condition. It simulates the mirrors by taking radiation into account on all boundaries and applying a low emissivity. The inner boundaries of the PV cells and of the insulation also make use of radiation boundary conditions. However, the PV cells have a high emissivity and the insulation a low emissivity. Further, the PV cells convert a fraction of the irradiation to electricity instead of heat. Heat sinks on their inner boundaries simulate this effect according to

$$q = -G\eta_{pv}$$

where G is the irradiation flux (W/m^2) and η_{pv} is the PV cell's voltaic efficiency. The latter depends on the local temperature, with a maximum of 0.2 at 800 K:

$$\eta_{pv} = \begin{cases} 0.2\left[1-\left(\frac{T}{800\ \mathrm{K}}-1\right)^2\right] & T \le 1600\ \mathrm{K} \\ 0 & T > 1600\ \mathrm{K} \end{cases}$$

Figure 2-57 illustrates this expression for temperatures above 1000 K.

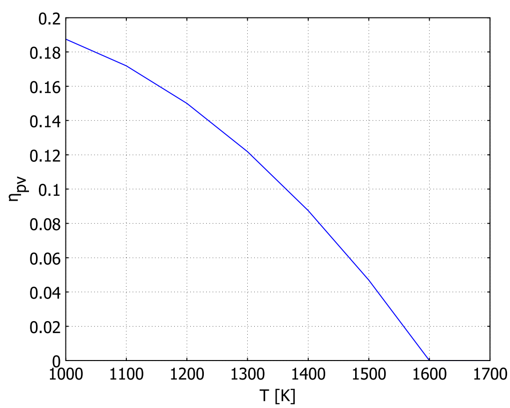


Figure 2-57: PV cell voltaic efficiency versus temperature.

At the outer boundary of the PV cells, the model applies convective water cooling by setting h to 50 W/(m^2·K), and T_{amb} to 273 K. Finally, at the outer boundary of the insulation it applies convective cooling with h set to 5 W/(m^2·K) and T_{amb} to 293 K.

Table 2-7 summarizes the material properties.

TABLE 2-7: MATERIAL PROPERTIES

COMPONENT	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]	ε
emitter	10	2000	900	0.99
mirror	10	5000	840	0.01
PV cell	93	2000	840	0.99
insulation	0.05	700	100	0.1

The model calculates the stationary solution for a range of emitter temperatures (1000 K to 2000 K) using the parametric solver.

Results and Discussion

The results shows that the device experiences a significant temperature distribution that varies with operating conditions. Figure 2-58 depicts the stationary distribution at operating conditions with an emitter temperature of 2000 K.

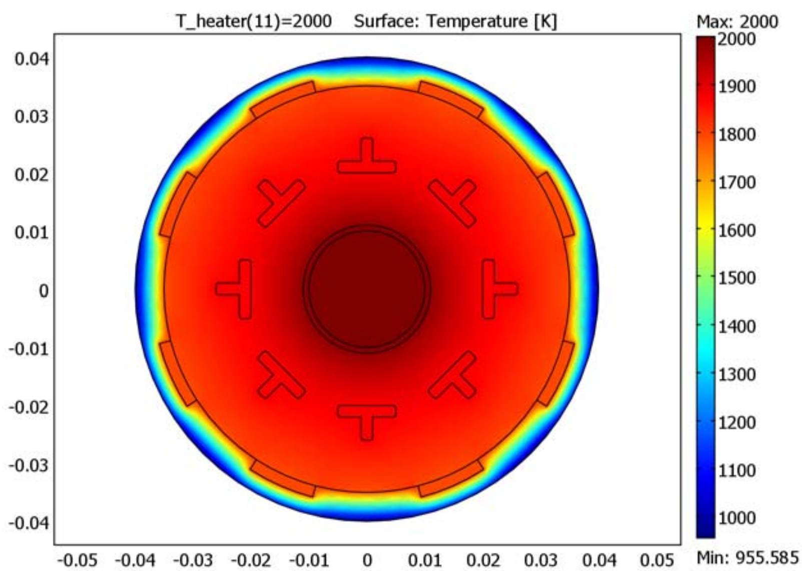


Figure 2-58: Temperature distribution in the TPV system when the emitter temperature is 2000 K.

As the upper plot in Figure 2-59 shows, the PV cells reach a temperature of approximately 1800 K. This is significantly higher than their maximum operating temperature of 1600 K, above which their photovoltaic efficiency is zero (see Figure 2-57 on page 147).

It is interesting to investigate what the optimal operating temperature is. The lower plot in Figure 2-59 investigates at what temperature the system achieves the maximum electric power output. The optimal emitter temperature for this configuration seems to be between 1600 K and 1700 K, where the electric power (irradiation multiplied by voltaic efficiency) is maximum.

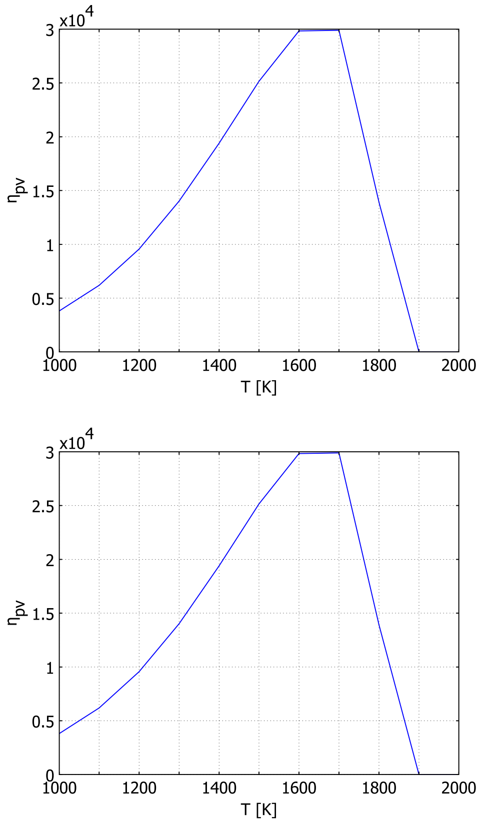


Figure 2-59: PV cell temperature (top) and electric output power (bottom) versus operating temperature.

The next step is to look at the temperature distribution at the optimal operating conditions (Figure 2-60).

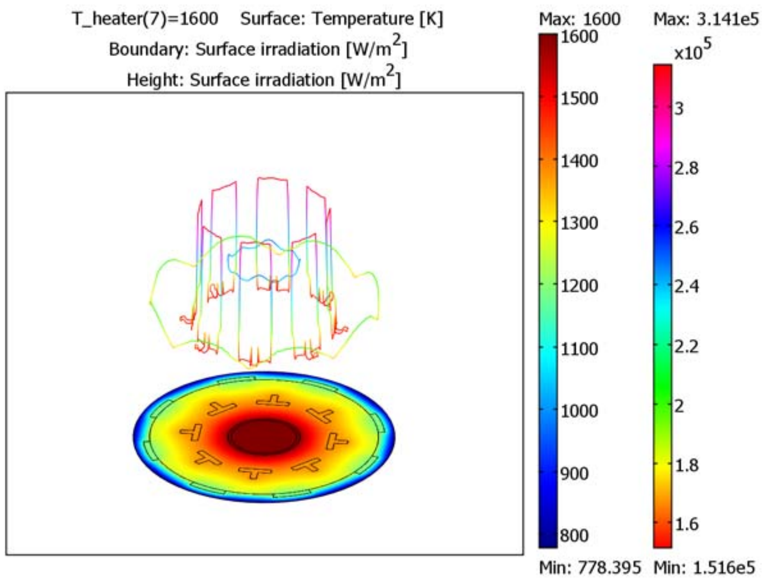


Figure 2-60: Temperature distribution and surface irradiation flux in the system at an operating emitter temperature of 1600 K.

When the emitter is at 1600 K, the PV cells reach a temperature of approximately 1200 K, which they can withstand without any problems. Note that the insulation reaches a temperature of approximately 800 K on the outside, suggesting that the system transfers a significant amount of heat to the surrounding air.

The plot also depicts the irradiative flux, which varies significantly along the circumference of the PV cell and insulation jacket. To further investigate this effect, Figure 2-61 plots the irradiative flux along a quarter of the circumference separately at this operating condition. Clearly the variation it shows is related to the positions of the mirrors and is an effect of shadowing.

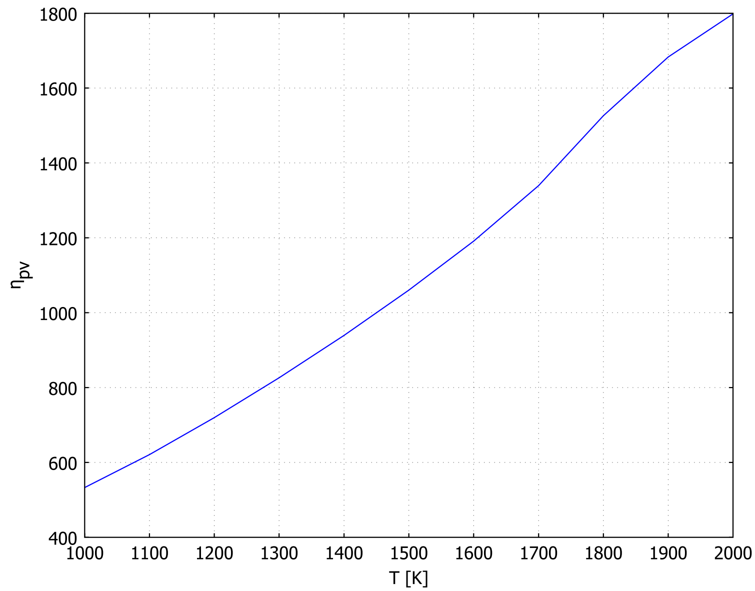


Figure 2-61: Irradiation flux along the PV cell and insulation inner surface for one quarter of the device circumference.

This plot can help optimize the mirror geometry as well as help decide how large the PV cells should be and where they should be placed.

A general conclusion is that this type of modeling can shortcut the prototype development time and optimize the operating conditions for the finalized TPV device.

References

1. http://lmn.web.psi.ch/shine/Flyer_TPV_E.pdf.

2. http://vri.etec.wwu.edu/viking_29_paper.htm.

3. Courtesy of E. Fontes, Catella Generics AB, Sweden.

4. Courtesy of Dr. D. Wilhelm, Paul Sherrer Institute, Switzerland.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/TPV_cell`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**. From the **Space dimension** list, select **2D**.

2 In the list of application modes, select **Heat Transfer Module>General Heat Transfer**.

3 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu open the **Axes/Grid Settings** dialog box. Go to the **Axis** page. In both the **x min** and **y min** edit fields, type `-0.05`, and in both the **x max** and **y max** edit fields, type `0.05`.

2 On the **Grid** page, clear the **Auto** check box. In both the **x spacing** and **y spacing** edit fields type `0.002`. Click **OK**.

3 From the **Options** menu, select **Constants**. In the dialog box that opens, enter the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_heater	1000[K]	Temperature, emitter inner boundary
Cp_air	1100[J/(kg*K)]	Specific heat capacity, air
h_air	5[W/(m^2*K)]	Heat transfer coefficient, air
T_air	293[K]	Temperature, air
k_ins	0.05[W/(m*K)]	Thermal conductivity, insulation
rho_ins	700[kg/m^3]	Density, insulation
Cp_ins	100[J/(kg*K)]	Specific heat capacity, insulation
e_ins	0.1	Surface emissivity, insulation
k_m	10[W/(m*K)]	Thermal conductivity, mirror
rho_m	5000[kg/m^3]	Density, mirror
Cp_m	840[J/(kg*K)]	Specific heat capacity, mirror
e_m	0.01	Surface emissivity, mirror
k_emit	10[W/(m*K)]	Thermal conductivity, emitter
rho_emit	2000[kg/m^3]	Density, emitter

NAME	EXPRESSION	DESCRIPTION
Cp_emit	900[J/(kg*K)]	Specific heat capacity, emitter
e_emit	0.99	Surface emissivity, emitter
k_pv	93[W/(m*K)]	Thermal conductivity, PV-cell
rho_pv	2000[kg/m^3]	Density, PV-cell
Cp_pv	840[J/(kg*K)]	Specific heat capacity, PV-cell
e_pv	0.99	Surface emissivity, PV-cell
h_cool	50[W/(m^2*K)]	Heat transfer coefficient, cooling water
T_cool	273[K]	Temperature, cooling water

4 Choose **Options>Expressions>Scalar Expressions**, then define the following expressions:

NAME	EXPRESSION
rho_air	1.013e5[Pa]*28.8e-3[kg/mol]/(8.31[J/(mol*K)]*T)
k_air	10^(-3.723+0.865*log10(abs(T[1/K])))[W/(m*K)]
eta_pv	0.2*(1-(T/800[K]-1)^2)*(T<1600[K])

5 Optionally, enter the descriptions `Density, air`; `Thermal conductivity, air`; and `Voltaic efficiency, PV cell`.

6 Click **OK**.

GEOMETRY MODELING

1 Create two circles. To do so, choose **Draw>Specify Objects>Circle**. For the first circle, type `0.035` in the **Radius** edit field, and for the second, type `0.037`.

2 Click the **Create Composite Object** button on the Draw toolbar. In the **Set formula** edit field, type `C2-C1`, then click **OK**.

3 Specify a rectangle by pressing Shift and clicking the **Rectangle** button on the Draw toolbar. In both the **Width** and **Height** edit fields, type `0.05`, and in the **x-base** edit field, type `-0.05`. Click **OK**.

4 Select all the objects and click the **Intersection** button on the Draw toolbar. This creates the composite object CO2, which is a quarter of an annulus.

5 To create the first mirror, create two rectangles: For each rectangle, press Shift then click the **Rectangle** button on the Draw toolbar, enter settings from the table below, and then click **OK**.

OBJECT	WIDTH	HEIGHT	BASE	X-BASE	Y-BASE
R1	`0.01`	`0.002`	Center	`-0.014`	`0.015`
R2	`0.002`	`0.006`	Center	`-0.014`	`0.017`

6 Select both rectangles, then click the **Create Composite Object** button on the Draw toolbar. Clear the **Keep interior boundaries** check box, then click **OK** to create the union.

7 From the **Draw** menu open the **Fillet/Chamfer** dialog box.

8 In the **Vertex selection** list click the **CO1** folder, then select Vertices 1, 2, 5, and 8–10. In the **Radius** edit field, type `0.5e-3`, then click **OK**. This step creates the composite object CO3.

9 Copy and paste CO3 by pressing Ctrl+C and then Ctrl+V. When pasting, specify the **Displacement** for **x** as `0.014` and for **y** as `0.006`. Click **OK**.

10 Select CO3. Click the **Rotate** button on the Draw toolbar. In the **Rotation angle** edit field type `45`. Go to the **Center point** area; in the **x** edit field type `-0.014`, and in the **y** edit field type `0.014`. Click **OK**.

11 Click the **Line** button on the Draw toolbar. Draw a line by left-clicking at (0, 0)—you can read the position in the lower left corner of the user interface—and at (–0.046, 0.012). Finalize the line by right-clicking in the drawing area.

12 Repeat the procedure in the previous step to draw three additional lines between the coordinates (0, 0) and (–0.024, 0.040), between (0, 0) and (–0.012, 0.046), as well as between (0, 0) and (–0.04, 0.024).

13 Select the circle object CO2 and the first line, B1. Click the **Coerce to Solid** button on the Draw toolbar.

14 Using the mouse, select the annulus object (now named CO4) and the second line. Click **Coerce to Solid**.

15 Select the annulus object (now named CO2) and the third line, B3. Click **Coerce to Solid**.

16 Select all objects (press Ctrl+A) and click **Coerce to Solid**.

17 Press Ctrl+C to copy the new composite object. Press Ctrl+V, then click **OK** to paste it with zero displacement.

18 Click **Rotate**. For the **Rotation angle** specify `90`, then click **OK**.

19 Repeat the paste-and-rotate procedure for `180` and `270` degrees to complete the circular object. Press Ctrl+D to clear the selection before selecting the original composite object to make a copy and then rotate it.

20 Draw two circles using the menu item **Draw>Specify Objects>Circle**. For the first one, type `0.04` in the **Radius** edit field, and for the second one, type `0.037`.

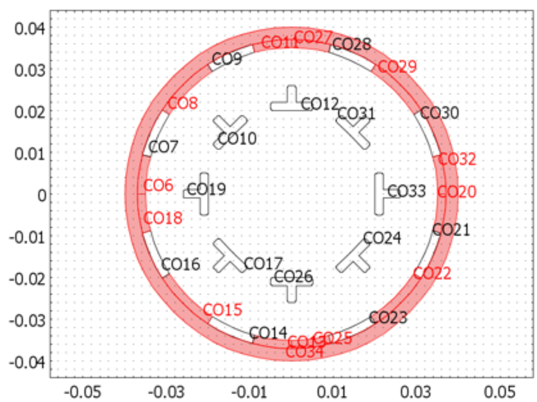
21 Click the **Zoom Extents** button on the Main toolbar.

22 Select the two circles, click the **Create Composite Object** button on the Draw toolbar, and in the **Set formula** edit field type the expression `C1-C2`. Click **OK**.

23 Select all objects (press Ctrl+A).

24 Click the **Split Object** button on the Draw toolbar.

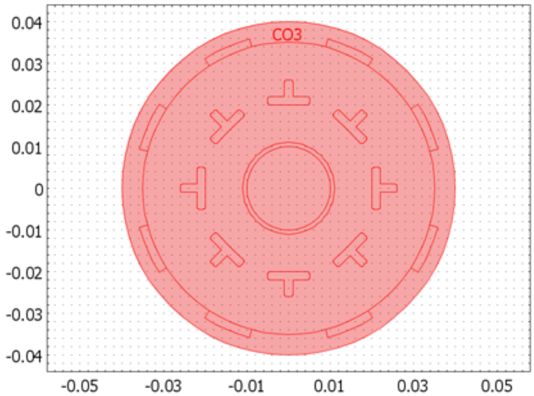
25 Select the objects indicated in the following figure, then click the **Create Composite Object** button on the Draw toolbar. Clear the **Keep interior boundaries** check box, then click **OK**.



26 Draw two circles using the menu item **Draw>Specify Objects>Circle**. For the first one, type `0.01` in the **Radius** edit field, and for the second one, type `0.011`.

27 Click the **Create Composite Object** button on the Draw toolbar, then in the **Set formula** edit field type `C2-C1`. Click **OK**.

28 To finalize the geometry select all objects, then click **Coerce to Solid**.



PHYSICS SETTINGS

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 In the **Init** page, select all the subdomains. In the $\mathbf{T(t_0)}$ edit field enter `T_air`.

3 Go to the **General** page and specify the following settings; when done, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAINS 2, 3, 6, 7, 12, 13, 17, 18	SUBDOMAINS 5, 8, 9, 10, 11, 14, 15, 16	SUBDOMAIN 19	SUBDOMAINS 4, 20
k	`k_ins`	`k_pv`	`k_m`	`k_emit`	`k_air`
ρ	`rho_ins`	`rho_pv`	`rho_m`	`rho_emit`	`rho_air`
C_p	`Cp_ins`	`Cp_pv`	`Cp_m`	`Cp_emit`	`Cp_air`
Opacity	Opaque	Opaque	Opaque	Opaque	Transparent

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select the **Interior boundaries** check box to enable the specification of interior boundaries.

3 Select all the boundaries of the mirror objects by using the mouse to draw a box around each mirror. Press and hold the Ctrl key on the keyboard to add selections.

4 Click the **Boundary Condition** tab. In the **Boundary condition** list, select **Heat source/sink**. In the **Radiation type** list, select **Surface-to-surface**. In the ε edit field for **Surface emissivity**, type `e_m`.

5 In the **Boundary selection** list, choose 97, 98, 141, and 148 (the outer boundaries of the insulation). In the **Boundary condition** list, select **Heat flux**. In the **h** edit field for the **Heat transfer coefficient**, type `h_air`, and in the $\mathbf{T_{inf}}$ edit field for **External temperature**, type `T_air`. In the **Radiation type** list, select **Surface-to-ambient**. In the ε edit field, type `e_ins`, and in the $\mathbf{T_{amb}}$ edit field for **Ambient temperature**, type `T_air`.

6 In the **Boundary selection** list, choose 101, 102, 105, 106, 133, 134, 142, 147, 167, 168, 183, and 184 (the inner boundaries of the insulation). In the **Boundary condition** list, select **Heat source/sink**. In the **Radiation type** list, select **Surface-to-surface**. In the ε edit field, type `e_ins`.

7 In the **Boundary selection** list, choose 99, 100, 119, 120, 157, 158, 181, and 182 from the list (the outer boundaries of the cells). In the **Boundary condition** list, select **Heat source/sink**. In the **h** edit field, type `h_cool`, and in the $\mathbf{T_{inf}}$ edit field type `T_cool`.

8 In the **Boundary selection** list, choose 103, 104, 115, 116, 155, 156, 179, and 180 (the inner boundaries of the cells). In the **Boundary condition** list, select **Heat source/sink**. In the **Radiation type** list select **Surface-to-surface**. In the $\mathbf{q_0}$ edit field for **Heat source/sink**, type `-G_htgh*eta_pv`, and in the ε edit field type `e_pv`.

9 In the **Boundary selection** list, choose 127, 128, 143, and 146 (the outer boundaries of the emitter). In the **Boundary condition** list, select **Heat source/sink**. In the **Radiation type** list, select **Surface-to-surface**. In the ε edit field, type `e_emit`.

10 Finally, in the **Boundary selection** list, choose 131, 132, 144, and 145 (the inner boundaries of the emitter). In the **Boundary condition** list, select **Temperature**. In the $\mathbf{T_0}$ edit field for **Temperature**, type `T_heater`.

11 Click **OK**.

MESH

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box.

2 In the **Predefined mesh sizes** list select **Coarser**.

3 Go to the **Boundary** page. Using the mouse, select all boundaries of the mirrors and of the emitter's outer boundary. In the **Maximum element size** edit field type `1e-3`.

4 Select all inner boundaries of the insulation and the PV cells (101, 102, 103, 104, 105, 106, 119, 120, 133, 134, 142, 147, 155, 156, 167, 168, 179, 180, 183, and 184). In the **Maximum element size** edit field type `2e-3`. Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu open the **Solver Parameters** dialog box.
2 On the **General** page, find the **Solver** list and select **Parametric**. In the **Linear system solver** list select **Direct (UMFPACK)**. In the **Parameter name** edit field type `T_heater`, and in the **Parameter values** edit field type `1000:100:2000`.
3 On the **Parametric** page, select the **Manual tuning of parameter step size** check box. In the **Initial step size** edit field type `100`. Specify a **Minimum step size** of `25` and a **Maximum step size** of `100`.
4 Click the **Stationary** tab. In the **Maximum number of iterations** edit field type `50`.
5 Click **OK**.
6 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

Figure 2-58 is the default postprocessing plot. To reproduce the plots in Figure 2-59, follow these steps:

1 From the **Postprocessing** menu open the **Domain Plot Parameters** dialog box.
2 On the **General** page, clear the check box next to the label **Element refinement** (doing so disables automatic refinement), then in the corresponding edit field type `1`.
3 Go to the **Point** page. In the **Point selection** list, choose Point 6.
4 In the **Expression** edit field, type `G_htgh*eta_pv`, then click **Apply** to generate the upper plot.
5 Click the **General** tab. From the **Plot in** list, select **New figure**.
6 Return to the **Point** page. In the **Expression** edit field, type `T`.
7 Click **OK** to close the dialog box and generate the lower plot.

To generate Figure 2-60, follow these steps:

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.
2 On the **General** page, find the **Parameter value** list and select **1600**.
3 On the **Boundary** page, select the **Boundary plot** check box.
4 On the **Boundary Data** page, select **Surface irradiation** from the **Predefined quantities** list.
5 On the **Height Data** page, select the **Height data** check box.
6 From the **Predefined quantities** list, select **Surface irradiation**.
7 In the **Boundary color** area, select **hsv** from the **Colormap** list.

8 Click **OK** to close the dialog box and generate the plot.

9 To make the axes and the grid disappear, double-click the **AXIS** and **GRID** buttons on the status bar at the bottom of the user interface.

To reproduce Figure 2-61, follow these steps:

1 From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box.

2 On the **General** page, find the **Solutions to use** list and select **1600**. From the **Plot in** list, select **New figure**.

3 Click the **Line/Extrusion** tab, and in the **Predefined quantities** list select **Surface irradiation**.

4 In the **Boundary selection** list, choose 102, 104, 106, 116, and 134 (a quarter of the system's inner wall). Click **OK**.

Convective Cooling of a Potcore Inductor

Introduction

The inductor is a common component in a variety of different electrical devices. Its usage ranges from power transformation to measurement systems. In small devices with many components, such as in laptop computers, heat generation can be a problem and has to be accounted for in the design. This model describes the heat transfer in a potcore inductor that is cooled by convective cooling.

Model Definition

The problem is axisymmetric, so the model only requires two space dimensions. The following figure describes the model geometry:

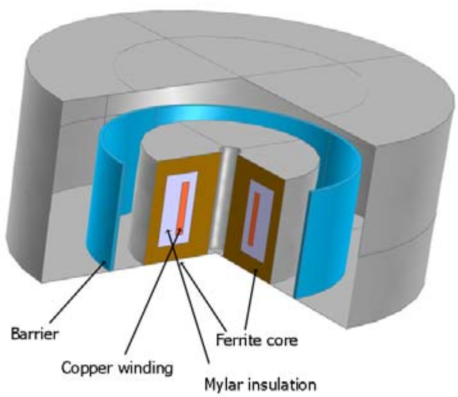


Figure 2-62: 3D view of the model geometry.

A varying current in the copper induces a magnetic field that is strengthened by the ferrite core. Heat is generated in the core and the winding due to resistive heating.

This model does not include the resistive heating due to induced currents, but instead assumes that a specific amount of heat is generated uniformly in the core and in the copper.

The component is cooled by air that enters from the top of the geometry and exits through the center and the lower part of the outer boundary.

Results and Discussion

Figure 2-63 shows the temperature distribution together with an arrow plot of the velocity field. The temperature has a a maximum in the copper winding where most of the heat is generated. It is clear that the air flow has a cooling effect on the temperature although this effect is not optimal. The arrow plot reveals that the air flow between the barrier and the ferrite core is very close to zero. Note also the recirculation zone in right part of the plot.

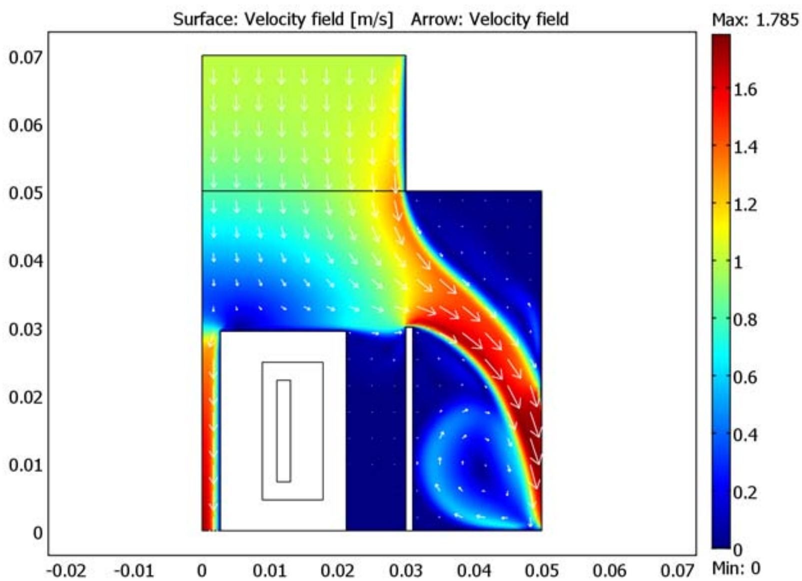


Figure 2-63: Surface plot of the temperature and arrow plot of the velocity field.

In the overall heat balance, radiation is responsible for about 10% of the total heat loss at steady state. The plot in Figure 2-64 shows a cross-sectional plot of the net radiative flux along the inner, vertical, boundary of the central hole (see Figure 2-62). Note that away from the open ends, the emitted and reflected radiation is almost balanced by the incident energy, so even if the temperature and radiation levels are high, the net flux is small in this region. The main part of the radiative losses instead take place from the outside of the inductor.

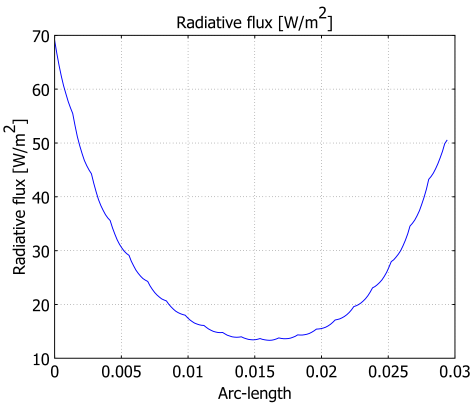


Figure 2-64: Cross-sectional plot of the net radiative flux.

Modeling in COMSOL Multiphysics

This model uses the General Heat Transfer application mode to solve for the temperature distribution. To provide cooling for the component, air enters the domain at the top of the geometry at the speed of 1 m/s. To include the air flow, the model also uses the Weakly Compressible Navier-Stokes application mode. The viscosity and density of air and hence the air flow depend on the temperature; on the other hand, the temperature distribution depends on the flow around the component. This means that this multiphysics model has to be solved simultaneously.

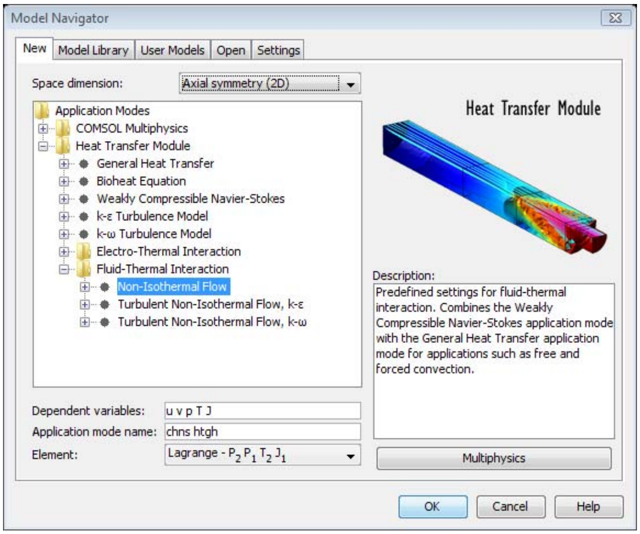
In this axisymmetric model, some of the surfaces are exposed to heat radiation from other surfaces, which means that surface-to-surface radiation must be accounted for. This type of radiation is quite complex because it depends on radiation from both the ambient and other surfaces. However, on some surfaces this complex boundary condition can be simplified to surface-to-ambient radiation. These are the surfaces that cannot be seen from any other radiating surfaces.

Model Library path: `Heat_Transfer_Module/`
`Electronics_and_Power_Systems/potcore_inductor`

MODEL NAVIGATOR

1 In the **Model Navigator**, select **Axial symmetry (2D)** from the **Space dimension** list.

2 Select **Heat Transfer Module>Fluid-Thermal Interaction>Non-Isothermal Flow.**



3 Click **OK** to close the **Model Navigator**.

OPTIONS AND SETTINGS

1 Open the **Constants** dialog box from the **Options** menu and enter the constants according to the following table (the descriptions are optional).

NAME	EXPRESSION	DESCRIPTION
Q_core	7.64e4[W/m^3]	Heat source in the core
Q_copper	8.657e5[W/m^3]	Heat source in the copper
p0	101.3[kPa]	Atmosphere pressure
T_amb	25[degC]	Ambient temperature
eps_ferrite	0.2	Surface emissivity of ferrite
eps_quartz	0.8	Surface emissivity of quartz

2 Click **OK** to close the dialog box.

GEOMETRY MODELING

1 Create rectangles according to the following table by shift-clicking on the **Rectangle/Square** button on the Draw toolbar and then specifying their width, height, and corner position.

WIDTH	HEIGHT	CORNER
0.05	0.05	(0,0)
0.03	0.02	(0,0.05)
0.0185	0.0294	(0.0027,0)
0.00895	0.0203	(0.00885,0.00455)
0.002	0.015	(0.011,0.0072)
0.001	0.03	(0.03,0)

2 Click the **Zoom Extents** button on the Main toolbar to fit the model geometry to your window.

3 Draw a line from (0.05, 0) to (0.05, 0.02).

The geometry should now look like that in Figure 2-65.

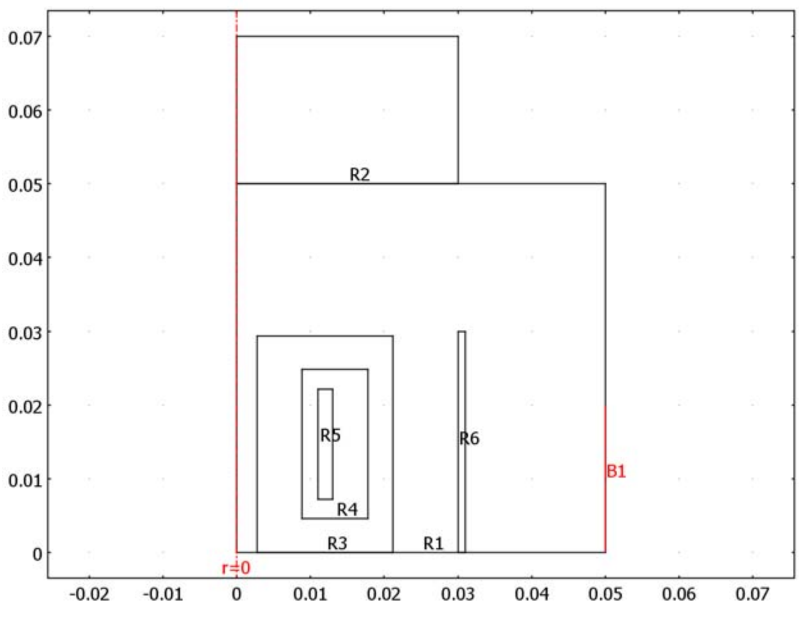


Figure 2-65: The model geometry.

PHYSICS SETTINGS

Subdomain Settings—General Heat Transfer

1 Select the **General Heat Transfer** application mode from the **Model Tree**.

2 From the **Physics** menu, choose **Subdomain Settings**.

3 Select Subdomains 1 and 2, then click the **Load** button.

4 Select **Air** from the **Basic Material Properties** library, then click **OK**.

Some of the boundaries in the model are exposed to surface-to-surface radiation. Before specifying this boundary condition you must first specify which subdomains that are opaque and which are transparent. By default, all subdomains are assumed to be opaque.

5 Select **Transparent** from the **Opacity** list.

6 Click the **Convection** tab.

7 Edit the field for the **Absolute pressure** so that it reads `p`.

8 Return to the **General** page.

9 Select Subdomains 3 through 6 and from the **Group** list, select **Solid domain**.

10 Select Subdomain 5, then click the **Load** button.

11 Select **Copper** from the **Basic Material Properties** library, then click **OK**.

12 Type `Q_copper` in the **Heat source** edit field.

13 For the remaining subdomains, specify the material properties according to the following table:

SETTINGS	SUBDOMAIN 3 (FERRITE)	SUBDOMAIN 4 (MYLAR)	SUBDOMAIN 6 (QUARTZ)
k (isotropic)	5	0.2	6.1
ρ	4800	1393	2648
C_p	750	1000	759

14 Select the ferrite subdomain (Subdomain 3), then type `Q_core` in the **Heat source** edit field.

15 Click **OK**.

Boundary Conditions—General Heat Transfer

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select the **Interior boundaries** check box, then specify the boundary conditions according to the following tables; when done, click **OK**.

SETTINGS	BOUNDARIES 1, 3	BOUNDARIES 5, 22, 23, 27	BOUNDARIES 2, 26	BOUNDARIES 7, 18, 20, 25
Boundary condition	Axial symmetry	Temperature	Convective flux	Insulation/ Symmetry
T_0		`T_amb`		
Radiation type		None		

SETTINGS	BOUNDARIES 6, 17	BOUNDARY 8	BOUNDARIES 21, 24	BOUNDARY 19
Boundary condition	Heat source/ sink	Heat source/ sink	Heat source/ sink	Heat source/ sink
T_{amb}	`T_amb`	`T_amb`	`T_amb`	`T_amb`
Radiation type	Surface-to-surface	Surface-to-surface	Surface-to-surface	Surface-to-surface
Surface emissivity	`eps_ferrite`	`eps_ferrite`	`eps_quartz`	`eps_quartz`

Subdomain Settings—Weakly Compressible Navier-Stokes

1 Select the **Weakly Compressible Navier-Stokes** application mode from the **Model Tree**.

2 From the **Physics** menu, open the **Subdomain Settings** dialog box.

The solid domains can be deactivated because there is no flow in them.

3 Select Subdomains 3–6, then from the **Group** list, select **Solid domain**.

4 Select Subdomains 1 and 2, then select **Air** from the **Library material** list.

5 On the **Init** page, type `p0` in the **Pressure** edit field.

6 On the **Stabilization** page, enter `0.3` in the **Tuning parameter** edit field for **Crosswind diffusion**.

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

The extra crosswind diffusion is needed to deal with the shear layers that appears in this model.

Boundary Conditions—Weakly Compressible Navier-Stokes

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Specify boundary conditions according to the following table; when done, click **OK**.

SETTINGS	BOUNDARIES 1, 3	BOUNDARY 5	BOUNDARY 2	BOUNDARY 26
Boundary type	Symmetry boundary	Inlet	Outlet	Outlet
Boundary condition	Axial symmetry	Velocity	Pressure, no viscous stress	No viscous stress
U_0		`1[m/s]`		
p_0			p0	

3 From the **Physics** menu, select **Point settings**.

4 Select Point 22, then select the **Point constraint** check box.

5 In the $\mathbf{P_0}$ edit field, type `p0`, then click **OK**.

MESH GENERATION

1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.

2 On the **Global** page, select **Extra fine** from the **Predefined mesh sizes** list.

3 On the **Subdomain** page, select subdomains 1 and 2.

4 From the **Method** list, select **Triangle**.

5 Click **Remesh** to create the mesh and then **OK** to close the dialog box.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar to compute the solution.

POSTPROCESSING AND VISUALIZATION

The default plot shows the temperature distribution. To create Figure 2-63, follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar.

2 Click the **Arrow** tab.

3 Select the **Arrow plot** check box.

4 From the **Predefined quantities** list, select **Weakly Compressible Navier-Stokes (chns)>Velocity field**.

5 In the **z points** edit field, type `18`.

6 Clear the **Auto** check box for **Scale factor**, and type `1.2` in the associated edit field.

7 Click the **Color** button. Select white from the palette, then click **OK**.

8 Click **OK**.

To view a cross-sectional plot of the surface irradiation, proceed with the following steps:

1. From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box.
2. Go to the **Line/Extrusion** page and select **General Heat Transfer (htgh)>Radiative flux** from the **Predefined quantities** list.
3. Select Boundary 6, then click **OK** to generate the plot in Figure 2-64.

Temperature Distribution in a Disc-Type Transformer

Introduction

This example illustrates a multiphysics application that involves heat transfer and fluid flow. The model simulates the steady-state temperature distribution in an oil-cooled ring-shaped transformer. It is based on published work by J.-M. Mufuta and others (Ref. 1).

Thermal aspects have a great importance in the design of large power transformers. First, sufficient cooling is necessary to avoid overheating. Second, the ageing of electrically insulating materials in transformers is directly proportional to the increase above a certain temperature. In order to design a transformer properly, it is necessary to study both the overall cooling power and the temperature distribution, which reveals where hot spots appear. They are a limiting factor in terms of ageing.

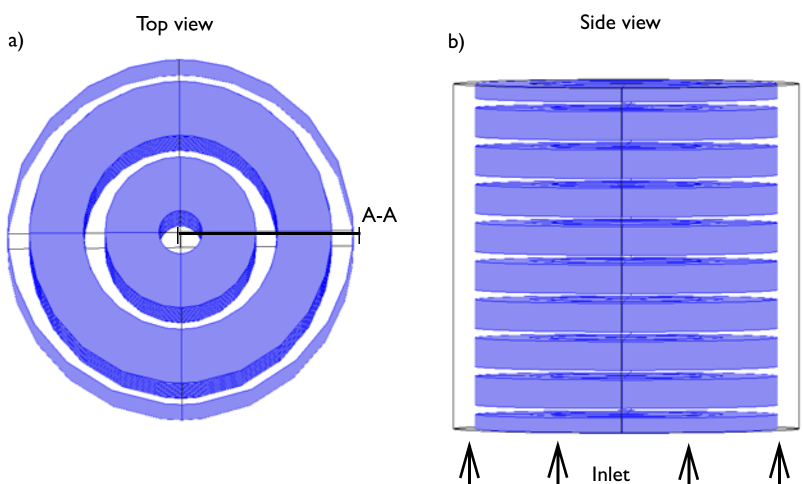


Figure 2-66: Geometry of the transformer coils.

The metallic transformer coils (Figure 2-66) heat up during operation. Transformer oil pumped through the coils perform the necessary cooling. The oil has a viscosity and density that vary with temperature, so heating affects the fluid-flow pattern. The model in this example simulates the steady-state temperature distribution in the transformer by modeling both the conduction-convection problem and the nonisothermal flow field. The geometry is axisymmetric, and this example models a unit cell consisting of 20 coils divided in two rows.

Model Definition

The model uses two stationary application modes to simulate the problem: Weakly Compressible Navier-Stokes and General Heat Transfer.

It simulates the momentum transport and mass conservation with the Weakly Compressible Navier-Stokes equations that describe the fluid velocity, $\mathbf{u}$, and the pressure field, p. In this case, the density, ρ, and the viscosity, η, are temperature dependent:

$$\rho\mathbf{u}\cdot\nabla\mathbf{u} = \nabla\cdot[-p\mathbf{I}+\eta(\nabla\mathbf{u}+(\nabla\mathbf{u})^T)-(2\eta/3)(\nabla\cdot\mathbf{u})\mathbf{I}]+\rho\mathbf{g}$$
$$\nabla\cdot(\rho\mathbf{u}) = 0$$

Variations in density result in buoyancy forces, expressed as $\rho\mathbf{g}$, and a continuity equation for the total mass, as expressed in the previous equations.

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

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

Here k represents thermal conductivity, C_p is the (temperature-dependent) specific heat capacity, and Q is the heating power per unit volume. For this model, use the value 32,400 W/m^3 for Q. Furthermore, set the thermal conductivities for the oil and the conductor material to 0.125 W/(m·K) and 383 W/(m·K), respectively.

The temperature-dependent expressions for ρ, η, and C_p used in the model read (this information comes from the producer of the transformer oil, 10GBN: Nynäs Petroleum AB, Stockholm, Sweden):

$$\rho = 875.6-0.63T \ \text{kg/m}^3$$

$$\eta = \rho 10^{(-4.726-0.0091T)} \ \text{m}^2\text{/s}$$

$$C_p = 1960+4.005T \ \text{J/(kg·K)}$$

In these expressions, T refers to the temperature value in degrees Celsius.

The transformer's cylindrical geometry allows for 2D axisymmetric modeling of a cross section as in Figure 2-67. The conductor coils are 25 mm wide and 15 mm high in cross section. The rows of coils have a radial separation of 10 mm and a vertical separation of 5 mm. The first row has a distance of 5 mm from the center. The gap between the second row and the outer wall is 10 mm.

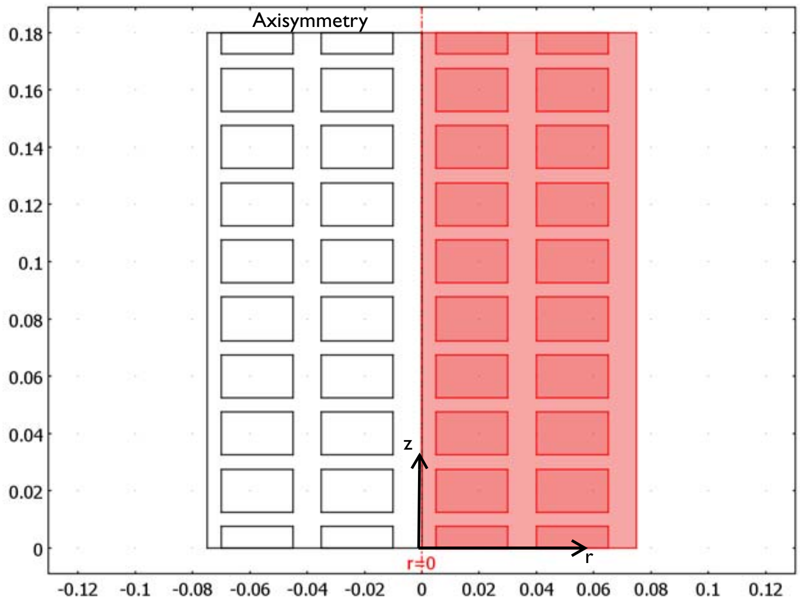


Figure 2-67: Transformer cross section. Because of axisymmetry you can use cylindrical coordinates and only need to include the shaded half in the model geometry.

At the boundary in the center of the cylinder, the model uses the axisymmetry condition for both application modes.

The fluid flow application boundary conditions are as follow. At the bottom boundary (the inlet), the fluid velocity is 5 mm/s in the z direction. At the top boundary (the outlet), the pressure is constant, and the r-velocity is zero. On the outer wall and on all coil surfaces, the fluid velocity is zero (no slip).

The boundary conditions for the heat equation application mode are:

- Oil inlet temperature of 50 °C
- Only convective heat flux at the outlet boundaries: $\mathbf{n} \cdot (-k\nabla T) = 0$
- Axisymmetry at the center ($r = 0$)

- Insulation at the outer wall ($r = 0.075$ m)
- The outer boundaries of the top and bottom conductor are thermally insulated

The solution of the specified equations and boundary conditions gives the temperature and flow field in the transformer.

Results and Discussion

One interesting result from this simulation concerns the temperature of the hot spot. Figure 2-68 depicts the temperature distribution at steady state for both the nonisothermal and an isothermal flow model, neglecting the variation in viscosity and density.

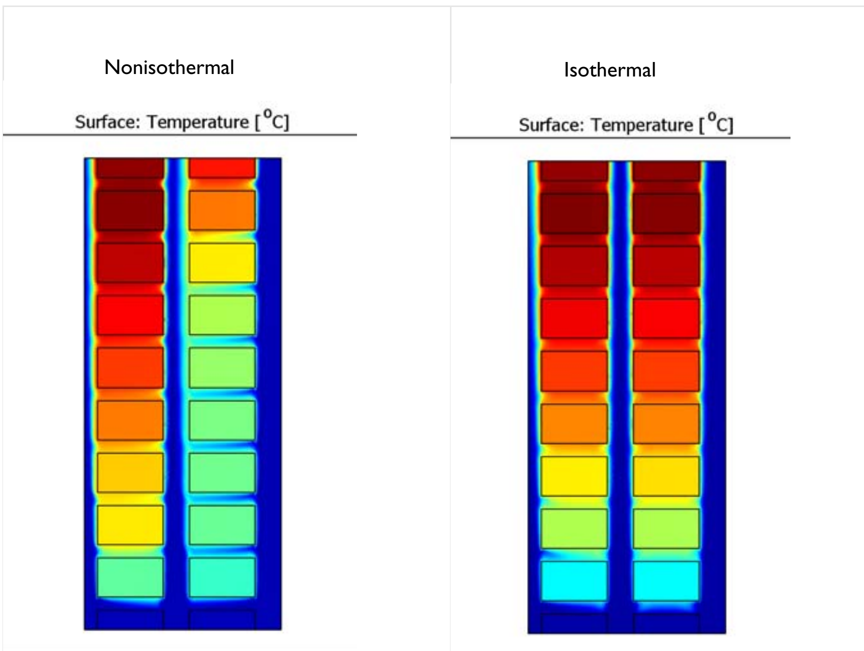


Figure 2-68: Temperature distribution in the transformer cross section.

For the nonisothermal model, the maximum temperature (at the hot spot) is 55 °C, occurring at the top inner coil, and note that the isothermal-flow model predicts a somewhat higher temperature. Also, there is no difference between the temperatures of the two columns. You can explain the differences between the models with their different fluid flows, which are affected by the temperature change.

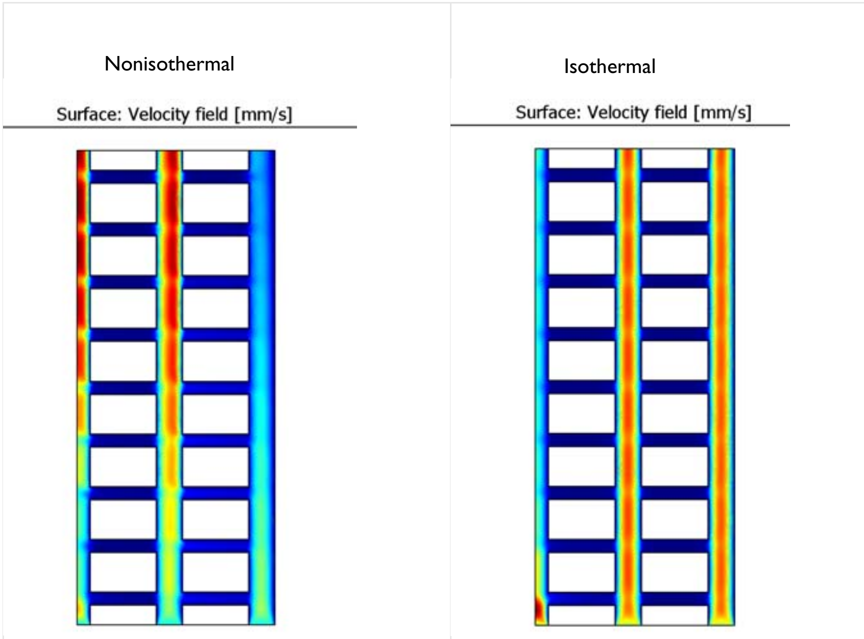


Figure 2-69: Fluid flow field in the transformer cross section.

Figure 2-69 shows the flow field for the two cases. The fluid velocity is higher for the non-isothermal case close to the center in the upper part of the transformer. This effect is caused mainly by the reduced viscosity due to a higher temperature. The buoyancy effects can also contribute to some extent. Furthermore, the flow field is more uniform between the vertical openings. In the isothermal case, the flow field is close to identical in the two outer vertical shafts. In this case, the velocity experiences a decrease in the central shaft due to the smaller shaft area.

In the non-isothermal model it is interesting to see that there exists a radial fluid velocity between the coils.

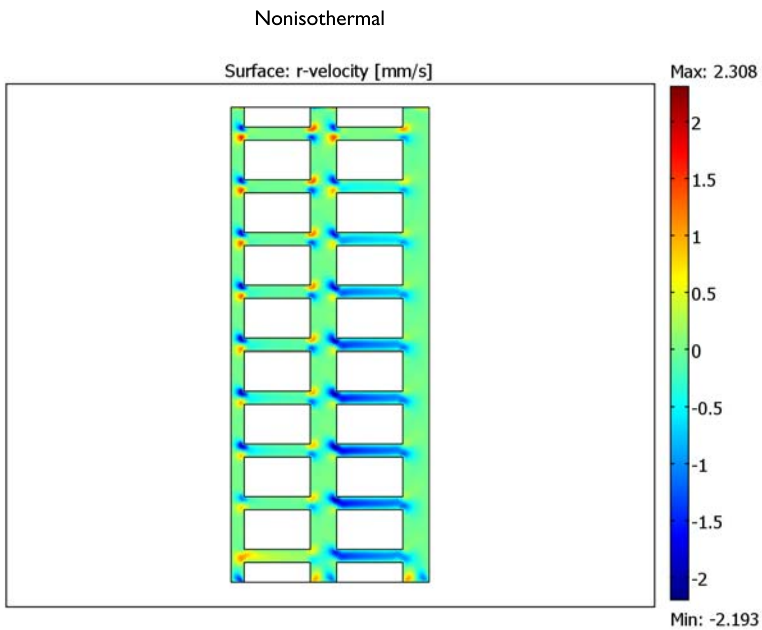


Figure 2-70: The radial component of the flow field in the nonisothermal case.

Figure 2-70 and Figure 2-71 display the radial velocity in the horizontal openings between the conductors. There is a flux of oil from the outer parts toward the middle of the transformer. The radial-velocity component varies in the transformer. Generally, the fluid flows toward the center of the transformer. The flow is more pronounced in the outer and lower regions.

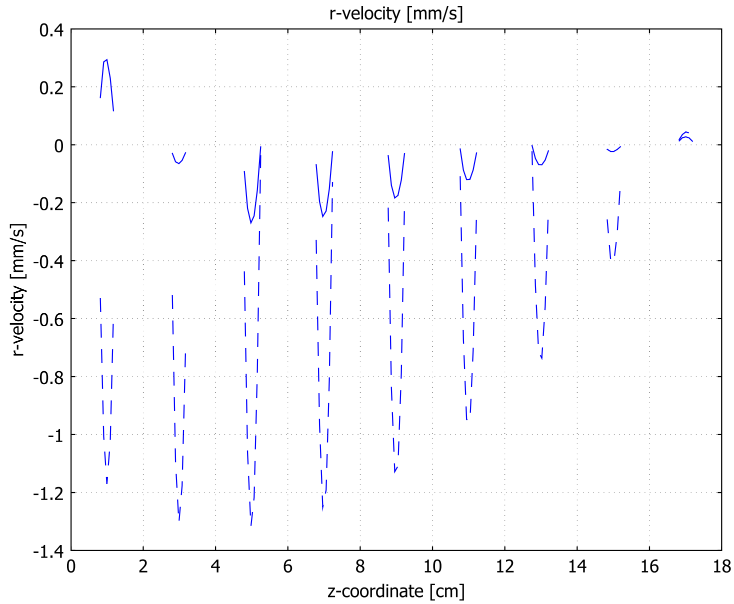


Figure 2-71: Radial fluid velocity for the non-isothermal model as a function of vertical position (z) at: r = 1.75 cm (solid line); and r = 5.25 cm (dashed line).

A general conclusion drawn from this model is that the variation in viscosity and density improve cooling. If you use modeling to optimize the transformer design with respect to hot spots, you should take the non-isothermal flow effects into account to produce more accurate simulation results.

Reference

1. J.-M. Mufuta and E. van Den Bulck, "Modeling of mixed convection in the windings of a disc-type power transformer," *Applied Thermal Engineering*, vol. 20, pp. 417–437, 2000.

Model Library path: `Heat_Transfer_Module/Electronics_and_Power_Systems/power_transformer`

MODEL NAVIGATOR

1 In the **Model Navigator**, go to the **Space dimension** list and select **Axial symmetry (2D)**.

2 In the list of application modes select **Heat Transfer Module>Fluid-Thermal Interaction>Non-Isothermal Flow**.

3 Click **OK**.

GEOMETRY MODELING

1 Press the Shift key and click the **Rectangle/Square** button on the Draw toolbar.

2 In the dialog box that appears enter these rectangle properties; when done, click **OK**.

PROPERTY	VALUE
Width	0.075
Height	0.18
Base	Corner
r	0
z	0

3 Click the **Zoom Extents** button on the Main toolbar.

4 Add a second rectangle, R2, with these properties:

PROPERTY	VALUE
Width	0.025
Height	0.0075
Base	Corner
r	0.005
z	0

5 With R2 selected, click the **Array** button on the Draw toolbar.

6 In the **Array** dialog box, enter the following settings; when done, click **OK**.

	DISPLACEMENT	ARRAY SIZE
r	0.035	2
z	0.1725	2

7 Add a rectangle, R6, with these properties:

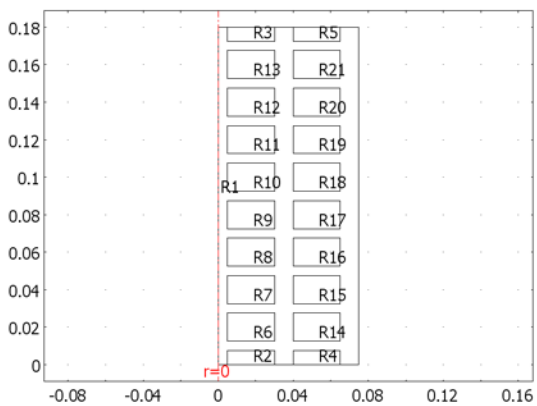
PROPERTY	VALUE
Width	0.025
Height	0.015
Base	Corner
r	0.005
z	0.0125

8 With R6 selected, click the **Array** button on the Draw toolbar.

9 In the **Array** dialog box, enter the following settings; when done, click **OK**.

	DISPLACEMENT	ARRAY SIZE
r	0.035	2
z	0.02	8

The geometry-modeling stage is now complete, with the following result:



OPTIONS AND SETTINGS

1 From the **Options** menu, select **Constants**.

2 Define the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T0	50[degC]	Inlet temperature, fluid
k_f	0.125[W/(m*K)]	Thermal conductivity, fluid

NAME	EXPRESSION	DESCRIPTION
k_s	383[W/(m*K)]	Thermal conductivity, solid
Q_s	32400[W/m^3]	Heating source
v0	5[mm/s]	Inlet flow velocity

PHYSICS SETTINGS

1 From the **Options** menu select **Expressions>Scalar Expressions**.

2 Define the following names and expressions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
rho	(875.6-0.63*T[1/degC])[kg/m^3]	Fluid density
Cp	(1960+4.0005*T[1/degC])[J/(kg*K)]	Specific heat capacity
eta	rho*10^(-4.726-0.0091*T[1/degC])[m^2/s]	Dynamic viscosity

Appending the operator `[1/degC]` to the variable `T` extracts the temperature value in degrees Celsius.

Subdomain Settings—Weakly Compressible Navier-Stokes

1 From the **Multiphysics** menu, select **Weakly Compressible Navier-Stokes (chns)**.

2 From the **Physics** menu, open the **Subdomain Settings** dialog box.

3 Choose Subdomains 2–21 (select **2** in the list, hold the Shift key down, then click number **21**). From the **Group** list, select **Solid domain**.

4 Select Subdomain 1.

5 On the **Physics** page, set η to `eta` and $\mathbf{F_z}$ to `-9.81[m^2/s]*rho`.

6 Click **OK**.

Subdomain Settings—General Heat Transfer

1 From the **Multiphysics** menu, select **General Heat Transfer (htgh)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select all subdomains. Click the **Init** tab, then type `T0` in the $\mathbf{T(t_0)}$ edit field.

4 Choose Subdomains 2–21 (select **2** in the list, hold the Shift key down, then click number **21**). From the **Group** list, select **Solid domain**.

5 Click the **General** tab.

6 Enter settings according to the following table (where the table entry is "-" leave the predefined setting); when done, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAINS 2–21
k (isotropic)	`k_f`	`k_s`
ρ	`rho`	-
C_p	`Cp`	0
Q	-	`Q_s`

Boundary Conditions—Weakly Compressible Navier-Stokes

1. From the **Multiphysics** menu, select **Weakly Compressible Navier-Stokes (chns)**.
2. From the **Physics** menu, open the **Boundary Settings** dialog box.
3. Select all boundaries by pressing Ctrl+A. In the **Boundary type** list, select **Wall**.
4. Adjust the boundary settings according to the following table; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARIES 2, 35, 77	BOUNDARIES 3, 45, 87
Boundary type	Symmetry boundary	Inlet	Outlet
Boundary condition	Axial symmetry	Velocity	Pressure, no viscous stress
v_0		v0	
p_0			0

Boundary Conditions—General Heat Transfer

1. From the **Multiphysics** menu, select **General Heat Transfer (htgh)**.
2. From the **Physics** menu, open the **Boundary Settings** dialog box.
3. Enter the following boundary conditions; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARIES 2, 35, 77	BOUNDARIES 3, 45, 87	BOUNDARIES 5, 33, 47, 75, 88
Boundary condition	Insulation/ Symmetry	Temperature	Convective flux	Insulation/ Symmetry
T_0		T0		

MESH GENERATION

1. From the **Mesh** menu, choose **Free Mesh Parameters**.
2. From the **Predefined mesh sizes** list, select **Coarser**.
3. Click the **Custom mesh size** button and type 4 in the **Resolution of narrow regions** edit field.

4 Click **OK**.

5 Click the **Initialize Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

For forced convection flows, it is often beneficial to use the stationary segregated solver.

1 Click the **Solver Parameters** button on the Main toolbar.

2 In the **Solver** list, select **Stationary segregated**, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To generate Figure 2-68, follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar.

2 Click the **Surface** tab. On the **Surface Data** page, select **General Heat Transfer (htgh)>Temperature** from the **Predefined quantities** list. Select **°C** from the **Unit** list, then click **Apply**.

To generate Figure 2-69 (the velocity field in the fluid), do this:

3 Still on the **Surface Data** page, change the entry in the **Predefined quantities** list to **Weakly Compressible Navier-Stokes (chns)>Velocity field**. Select **mm/s** from the **Unit** list, then click **Apply**.

To create Figure 2-70, follow this step:

4 Still on the **Surface Data** page, change the selection in the **Predefined quantities** list to **Weakly Compressible Navier-Stokes (chns)>r-velocity**. Select **mm/s** from the **Unit** list, then click **OK**.

To produce Figure 2-71 execute the following instructions:

1 From the **Postprocessing** menu open the **Cross-Section Plot Parameters** dialog box.

2 On the **Line/Extrusion** page, go to the **y-axis data** area and from the **Predefined quantities** list, select **Weakly Compressible Navier-Stokes (chns)>r-velocity**. From the **Unit** list, select **mm/s**.

3 In the **x-axis data** area, click first the lower option button and then the **Expression** button.

4 In the **X-Axis Data** dialog box, type `z` in the **Expression** edit field and `cm` in the **Unit** edit field. Click **OK** to close the dialog box.

5 Go to the **Cross-section line data** area. In both the **r0** and **r1** edit fields, type `0.0175`. In the **z0** edit field type `0`, and in the **z1** edit field type `0.18`. Click **Apply**.

6 On the **General** page, select the **Keep current plot** check box.

7 Return to the **Line/Extrusion** page, and type `0.0525` in both the **r0** and **r1** edit fields.

8 Click the **Line Settings** button. In the resulting dialog box, go to the **Line style** list and select **Dashed line**. Click **OK**.

9 Click **OK**.

To set up and solve the isothermal model, do the following:

1 From the **Options** menu, select **Expressions>Scalar Expressions**. In all the expressions in the table, replace the variable `T` with `T0`.

2 Repeat the steps from the section "Computing the Solution" to solve the new model. Repeat the steps from the section "Postprocessing and Visualization" to generate the result plots.

3

Processing and Manufacturing Models

In this chapter you find models that show heat transfer applications within the processing and manufacturing industries.

Heat Generation in a Disc Brake

Introduction

This example models the heat generation and dissipation in a disc brake of an ordinary car during panic braking and the following release period. As the brakes slow the car, they transform its kinetic energy into thermal energy, resulting in intense heating of the brake discs. If the discs overheat, the brake pads stop working and, in a worst-case scenario, can melt. Braking power starts to fade already at temperatures above 600 K.

In the model, the car (1800 kg) initially travels at 25 m/s (90 km/h) when the driver brakes hard for 2 s, causing the vehicle's eight brake pads to slow the car down at a rate of 10 m/s^2. The wheels are assumed not to skid against the road surface. After this period of time, the driver releases the brake and the car travels at 5 m/s for an additional 8 s without any braking. The questions to analyze with the model are:

- How hot do the brake discs and pads become during the braking stage?
- How much do they cool down during the subsequent rest?

Model Definition

Model the brake disc as a 3D (x, y, z) solid with shape and dimensions as in Figure 3-1. The disc has a radius of 0.14 m and a thickness of 0.013 m.

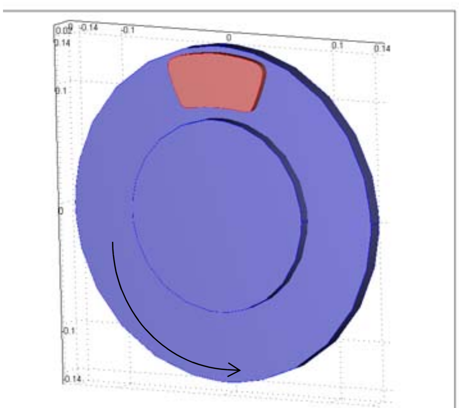


Figure 3-1: Geometry and dimensions of the modeled brake disc, including the brake pad.

Neglecting drag and other losses outside the brakes, the brakes' retardation power is given by the negative of the time derivative of the car's kinetic energy:

$$P = -\frac{d}{dt}\left(\frac{mv^2}{2}\right) = -mv\frac{dv}{dt} = -mR^2\omega(t)\alpha\,.$$

Here m is the car's mass, v denotes its speed, R equals the wheel radius (0.25 m), ω is the angular velocity, and α is the angular acceleration. The acceleration is constant in this case, so $\omega(t) = \omega_0 + \alpha t$.

By definition, the retardation power equals the negative of the work per unit time done by the friction forces on the discs at the interfaces between the pads and the discs for the eight brakes. You can calculate this work as eight times an integral over the contact surface of a single brake pad. The friction force per unit area, $\mathbf{f}_f$, is approximately constant over the surface and is directed opposite the disc velocity vector, $\mathbf{v}_d = v_d\mathbf{e}_\varphi$, where $\mathbf{e}_\varphi$ denotes a unit vector in the azimuthal (angular) direction and the magnitude of $\mathbf{v}_d$ at the distance r from the center equals $v_d(r, t) = \omega(t)r$. Thus, writing $\mathbf{f}_d = f_f\mathbf{e}_\varphi$ gives the following result for the retardation power:

$$P = -8\iint \mathbf{f}_f dA \cdot \mathbf{v}_d = 8f_f(t)\,\omega(t)\iint r dA$$

You can approximate the last integral with the pad's area, A (0.0035 m^2), multiplied by the distance from the center of the disc to the pad's center of mass, r_m (0.1143 m).

Combining the two expressions for P gives the following result for the magnitude of the friction force, f_f:

$$f_f = -\frac{mR^2\alpha}{8r_m A}$$

(Note that α is negative during retardation.)

Under the previously stated idealization that retardation is due entirely to friction in the brakes, the heat power generated per unit contact area at time t and the distance r from the center becomes

$$q(r, t) = -\mathbf{f}_f \cdot \mathbf{v}_d(r, t) = -\frac{mR^2\alpha}{8r_m A} r(\omega_0 + \alpha t)$$

The disc and pad dissipate the heat produced at the boundary between the brake pad and the disc by convection and radiation. This example models the rotation as convection in the disc. The local disc velocity vector is

$$\mathbf{v}_d = \omega(t)(-y, x)$$

The model also includes heat conduction in the disc and the pad through the transient heat transfer equation

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = Q - \rho C_P \mathbf{u} \cdot \nabla T$$

where k represents the thermal conductivity (W/(m·K)), C_p is the specific heat capacity (J/(kg·K)), and Q is the heating power per unit volume (W/m^3), which in this case is set to zero.

At the boundary between the disc and the pad, the brake produces heat according to the expression for q given earlier. The heat dissipation from the disc and pad surfaces to the surrounding air is described by both convection and radiation

$$q_{\mathrm{diss}} = -h(T - T_{\mathrm{ref}}) - \varepsilon\sigma(T^4 - T_{\mathrm{ref}}^4)$$

In this equation, h equals the convective film coefficient (W/(m^2·K)), ε is the material's emissivity, and σ is the Stefan-Boltzmann constant ($5.67 \cdot 10^{-8}$ W/(m^2·K^4)).

To calculate the convective film coefficient as a function of the vehicle speed, v, use the following formula (Ref. 1):

$$h = \frac{0.037k}{l} \mathrm{Re}^{0.8} \mathrm{Pr}^{0.33} = \frac{0.037k}{l} \left(\frac{\rho l v}{\mu}\right)^{0.8} \left(\frac{C_p \mu}{k}\right)^{0.33}$$

Here l is the disc's diameter. The material properties—the thermal conductivity, k, the density, ρ, the viscosity, μ, and the specific heat capacity, C_p—are those for air.

Table 3-1 summarizes the thermal properties, which come from (Ref. 1). You calculate the density of air at a reference temperature of 300 K using the ideal gas law.

TABLE 3-1: MATERIAL PROPERTIES

PROPERTY	DISC	PAD	AIR
ρ (kg/m^3)	7870	2000	1.170
C_p (J/(kg·K))	449	935	1100
k (W/(m·K))	82	8.7	0.026
ε	0.28	0.8	-
μ (Pa·s)	-	-	$1.8 \cdot 10^{-5}$

Results and Discussion

The surface temperatures of the disc and the pad vary with both time and position. At the contact surface between the pad and the disc the temperature increases when the brake is engaged and then decreases again as the brake is released. You can best see these results in COMSOL Multiphysics by generating an animation. Figure 3-2 displays the surface temperatures just before the end of the braking. A "hot spot" is visible at the contact between the brake pad and disc, just at the pad's edge—this is where the temperature could become critical during braking. The figure also shows the temperature decreasing along the rotational trace after the pad. During the rest, the temperature becomes significantly lower and more uniform in the disc and the pad.

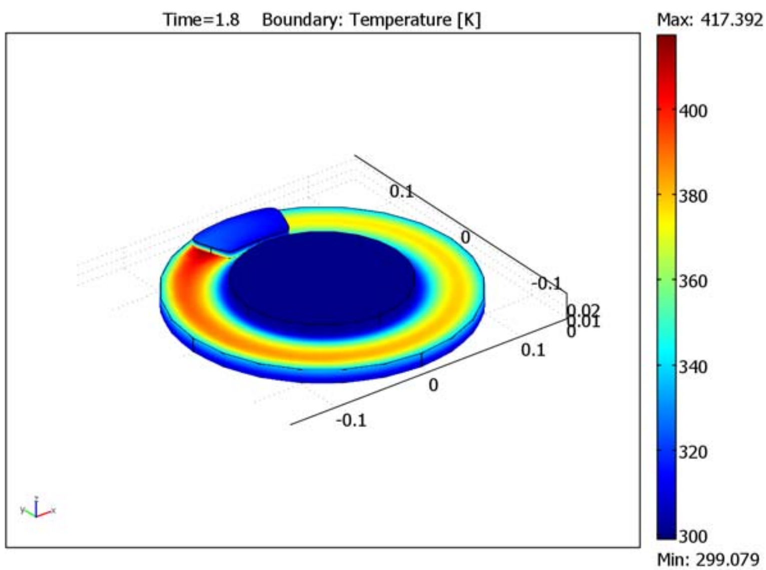


Figure 3-2: Surface temperature of the brake disc and pad just before releasing the brake ($t = 1.8$ s).

To investigate the position of the hot spot and the time of the temperature maximum, it is helpful to plot temperature versus time along a line from the center to the pad's edge as in Figure 3-3. You can see that the maximum temperature is approximately 440 K. The hot spot is positioned close to the radially outer edge of the pad. The highest temperature occurs approximately 1 s after engaging the brake.

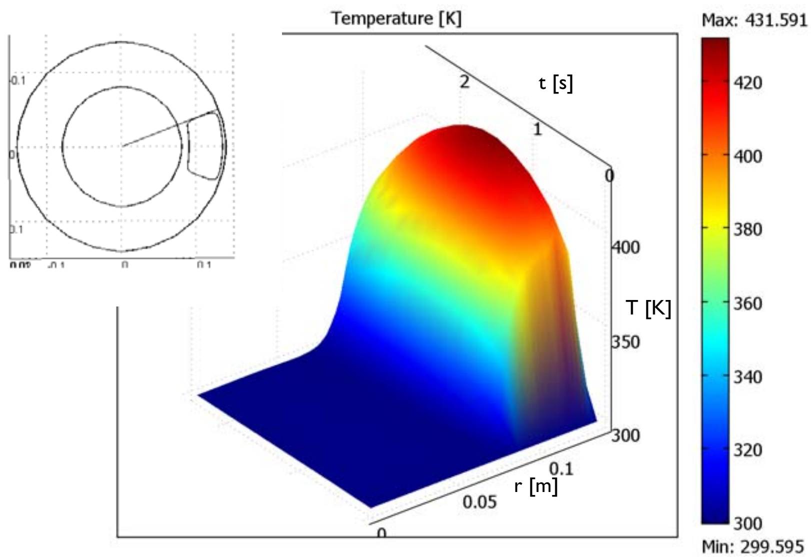


Figure 3-3: Temperature profile along the indicated line at the disc surface (z = 0.013 m) as a function of time.

To investigate how much of the generated heat is dissipated to the air, study the surface integrals of the produced heat and the dissipated heat. These integrals give the total heat flux (J/s) for heat production, Q_{prod}, and heat dissipation, Q_{diss}, as functions of time for the brake disc. The time integrals of these two quantities give the total heat (J) produced and dissipated, respectively, in the brake disc. Figure 3-4 shows a plot of the total produced heat and dissipated heat versus time. You can see that 8 s after disengagement the brake has dissipated only a fraction of the produced heat. The plot indicates that the resting time must be extended significantly in order to dissipate all the generated heat.

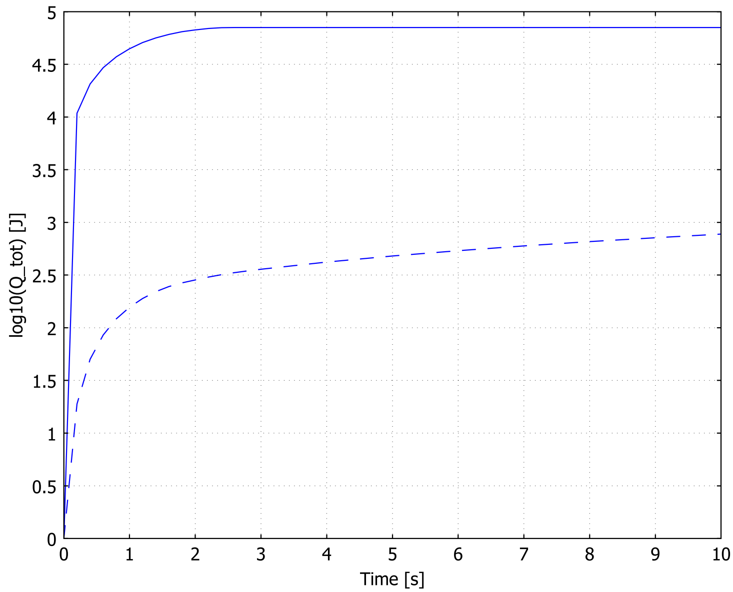


Figure 3-4: Comparison of total heat produced (solid line) and dissipated (dashed).

The results of this model can help engineers investigate how much abuse, in terms of specific braking sequences, a certain disc-break design can tolerate before overheating. It is also possible to vary the parameters affecting the heat dissipation and investigate their influence.

Reference

1. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, eq. 9.88; material properties from appendix A2.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/brake_disc`

MODEL NAVIGATOR

1 Open the **Model Navigator**. Click the **New** tab. From the **Space dimension** list select **3D**.

2 From the **Application Modes** tree, select **Heat Transfer Module> General Heat Transfer>Transient analysis**. Click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu, select **Constants**. Define the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
rho_disc	7870[kg/m^3]	Density, disc
rho_pad	2000[kg/m^3]	Density, pad
C_disc	449[J/(kg*K)]	Heat capacity, disc
C_pad	935[J/(kg*K)]	Heat capacity, pad
k_disc	82[W/(m*K)]	Thermal conductivity, disc
k_pad	8.7[W/(m*K)]	Thermal conductivity, pad
e_disc	0.28	Emissivity, disc
e_pad	0.8	Emissivity, pad
v0	25[m/s]	Initial vehicle speed
a0	-10[m/s^2]	Vehicle acceleration
r_wheel	0.25[m]	Wheel radius
omega0	v0/r_wheel	Initial angular velocity, disc
alpha	a0/r_wheel	Angular acceleration, disc
m_car	1800[kg]	Vehicle mass
A_pad	35e-4[m^2]	Surface area, pad
r_m	0.1143[m]	Center-of-mass radius, pad
f_f	-m_car*r_wheel^2*alpha/ (8*r_m*A_pad)	Friction force
t_brake	2[s]	Braking time
T_air	300[K]	Temperature, air
k_air	0.026[W/(m*K)]	Thermal conductivity, air
C_air	1100[J/(kg*K)]	Heat capacity, air
mu_air	1.8e-5[Pa*s]	Dynamic viscosity, air
rho_air	1.013e5[Pa]*28.8e-3[kg/mol]/ (8.314[J/(K*mol)]*T_air)	Density, air

GEOMETRY MODELING

1 Create two cylinders. To do so, click the **Cylinder** button on the Draw toolbar. Then enter settings from the following table. After creating each cylinder, click **OK**.

CYLINDER PARAMETER	CYL1	CYL2
Radius	0.14	0.08
Height	0.013	0.01
Axis base point x	0	0
Axis base point y	0	0
Axis base point z	0	0.013

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

3 Create a work plane along the disc surface. From the **Draw** menu select **Work-Plane Settings**. On the **Quick** page select the **x-y** option button, and in the a **z** edit field for the offset enter `0.013`. Click **OK**.

4 From the **Options** menu, open the **Axes/Grid Settings** dialog box.

5 On the **Axis** page and the **Grid** page, in turn, clear the **Auto** check box and enter settings from the following table; when finished, click **OK**.

AXIS		GRID	
x min	-0.07	x spacing	0.005
x max	0.07		
y min	0.08	y spacing	0.005
y max	0.15		

Next, draw the brake pad profile:

6 On the user interface, click the **Geom2** tab.

7 Click the **3rd Degree Bézier Curve** button on the Draw toolbar.

8 Draw a curve with the control points listed in the following table.

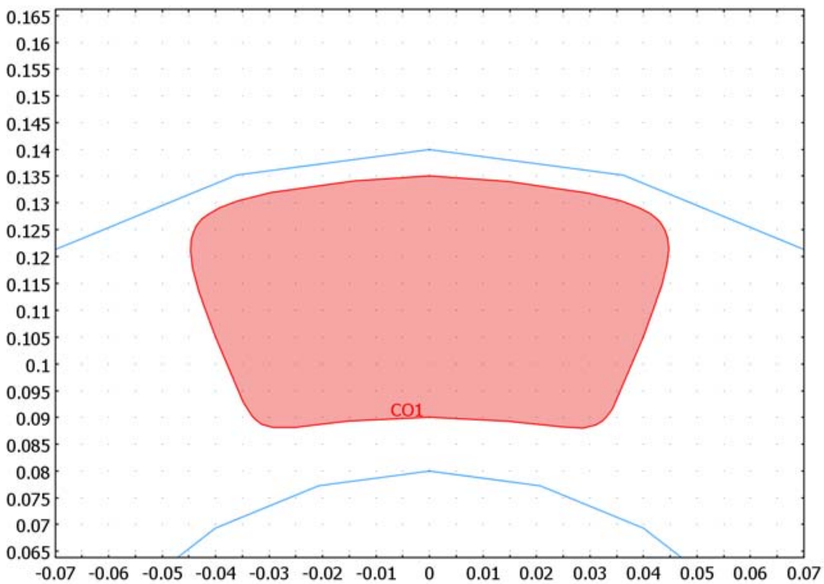
CONTROL POINTS
(0, 0.135), (0.02, 0.135), (0.05, 0.13), (0.04, 0.105), (0.03, 0.08), (0.035, 0.09), (0, 0.09), (-0.035, 0.09), (-0.03, 0.08), (-0.04, 0.105), (-0.05, 0.13), (-0.02, 0.135), (0, 0.135)

To find the points with the mouse, look at the coordinate indicator in the bottom left corner of the user interface. Mark each point by clicking the left mouse button. After the last point, click the right mouse button to close the curve and change it to a solid.

9 To complete the pad, you must make the top left and right corners sharper. Do so by changing the weights of the Bézier curves. From the **Draw** menu open the **Object Properties** dialog box, then change the weights for two of the curves using the information in the following table; when finished, click **OK**.

CURVE	POINT	WEIGHT
2	2	`2.5`
4	3	`2.5`

The geometry in the drawing area should now look like that in the figure below.



10 Make sure the object is selected, then from the **Draw** menu select **Extrude**. In the **Distance** edit field type `0.0065`, then click **OK**.

PHYSICS SETTINGS

1 From the **Options** menu, select **Expressions>Scalar Expressions**. Specify the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
v	(v0+a0*t)*(t<=t_brake)+ (v0+a0*t_brake)*(t>t_brake)	Vehicle speed
omega	v/r_wheel	Angular velocity, disc
h_air	0.037*k_air/(0.14[m]*2)* (rho_air*0.14[m]*2*v/mu_air)^0.8* (C_air*mu_air/k_air)^0.33	Convective film coefficient
q_prod	f_f*sqrt(x^2+y^2)*(omega0+alpha*t)* flc2hs((t_brake-t)[1/s],0.01)	Produced heat power per unit contact area
q_d_disc	h_air*(T_air-T)+e_disc* sigma_htgh*(T_air^4-T^4)	Dissipated heat power per unit disc area
q_d_pad	h_air*(T_air-T)+e_pad* sigma_htgh*(T_air^4-T^4)	Dissipated heat power per unit pad area

Here sigma_htgh is a predefined scalar variable for the Stefan-Boltzmann constant.

2 Make sure **Geom1** is the active page in the user interface.

3 Now set up some variables needed for postprocessing. From the **Options** menu, select **Integration Coupling Variables>Boundary Variables**. In the **Name** column, enter the two variables Dis_heat and Prod_heat. For certain boundaries, you must make an entry in the **Expression** column for one of those variables as specified in this table; when done, click **OK**.

SETTINGS	BOUNDARY 11	BOUNDARIES 1, 2, 4–6, 8, 13–15	BOUNDARIES 9, 10, 12, 16, 17
Dis_heat		-q_d_disc	-q_d_pad
Prod_heat	q_prod		

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select all the subdomains. Click the **Init** tab. In the **Temperature** edit field enter T_air.

3 Click the **General** tab. Enter the following settings:

SETTINGS	SUBDOMAINS 1, 2	SUBDOMAIN 3
k (isotropic)	k_disc	k_pad
ρ	rho_disc	rho_pad
C_p	C_disc	C_pad

4 Select Subdomains 1 and 2. Click the **Convection** tab. Select the **Enable convective heat transfer** check box.

5 From the **Matter state** list, select **Solid**.

6 In the leftmost **u** edit field for the *x* velocity type `-y*omega`, and in the middle edit field for the *y* velocity type `x*omega`.

7 Click the **Stabilization** tab. Click the **Elements of type** button; leave the default selection (**Lagrange - Quadratic**) in the corresponding list.

(The default solver is GMRES with geometric multigrid as preconditioner, but the problem is small enough to be solved with a direct solver).

8 Click **OK**.

Boundary Conditions

1 From the **Physics** menu, open the **Boundary Settings** dialog box. Select the **Interior boundaries** check box to enable boundary conditions on interior boundaries.

2 Specify the boundary conditions by entering the settings from this table; when done, click **OK**:

SETTINGS	BOUNDARY 3	BOUNDARIES 1, 2, 4–6, 8, 13–15, 18	BOUNDARY 11	BOUNDARY 9, 10, 12, 16, 17,
Type	Insulation/ Symmetry	Heat flux	Heat source/ sink	Heat flux
q_0		0	q_prod	0
h		h_air	0	h_air
T_{inf}		T_air	0	T_air
Radiation type	None	Surface-to-ambient	None	Surface-to-ambient
ε		e_disc		e_pad
T_{amb}		T_air		T_air

MESH GENERATION

1 From the **Mesh** menu, open the **Free Mesh Parameters** dialog box.

2 Click the **Boundary** tab. Select Boundary 4. In the **Maximum element size** edit field type `10e-3`. Similarly, select Boundary 11 and type `5e-3`.

3 Go to the **Advanced** page. In the **z-direction scale factor** edit field type `2`. Click **Remesh**, then click **OK**.

ADDITIONAL APPLICATION MODE

In order to integrate the heat produced and dissipated over time, this model uses a Weak Form, Point application mode.

1. From the **Multiphysics** menu, open the **Model Navigator**.
2. In the list of application modes select **COMSOL Multiphysics>PDE Modes>Weak Form, Point>Time-dependent analysis**.
3. Change the dependent variable names. In the **Dependent variables** edit field type `uP uD`. Click **Add**, then click **OK**.
4. From the **Physics** menu, open the **Point Settings** dialog box. Deactivate the application mode in all points; to do so, select all the points, then clear the **Active in this domain** check box.
5. Select Point 1, then select the **Active in this domain** check box to activate the Weak Form, Point application mode.
6. For Point 1, make the following entries in the edit fields on the **weak** and **dweak** pages; when done, click **OK**.

EDIT FIELD	POINT 1	POINT 2
weak	`uP_test*Prod_heat`	`uD_test*Dis_heat`
dweak	`uP_test*uP_time`	`uD_test*uD_time`

COMPUTING THE SOLUTION

1. From the **Solve** menu, open the **Solver Parameters** dialog box. On the **General** page, go to the **Times** edit field and type `0:0.2:3 4:10`.
2. In the **Linear system solver** list, select **Direct (UMFPACK)**, then click **OK**.
3. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the temperature at the last time step. Because the problem is time dependent, the natural way to view the solution is as an animation.

1. From the **Postprocessing** menu, open the **Plot Parameters** dialog box. In the **General** page go to the **Plot type** area. Clear the **Slice** check box, then select the **Boundary** check box.
2. Go to the **Animate** page, then click the **Start Animation** button. The animation appears in a separate window.

To create Figure 3-2, continue with this step:

1 While still working in the **Plot Parameters** dialog box, click the **General** tab.

2 In the **Solution at time** list, select **1.8**, then click **OK**.

To create Figure 3-3, which shows the temperature along a cross section as a function of time, continue with these steps:

1 From the **Postprocessing** menu, select **Cross-Section Plot Parameters**.

2 Go to the **General** page and find the **Solutions to use** area. Select all the solutions from 0 to 2.4 seconds.

3 Click the **Line/Extrusion** tab, go to the **Plot type** area, and select the **Extrusion plot** option button.

4 Go to the **x-axis data** area. In the list at the top of this area select **y**.

5 Still on the **Line/Extrusion** page, go to the **Cross-section line data** area. Enter settings as in the following table; when finished, click **OK**.

ENTRY	VALUE
x1	`-0.047`
y1	`0.1316`
z0	`0.013`
z1	`0.013`
Line resolution	`50`

To create Figure 3-4, continue with these steps:

1 From the **Postprocessing** menu, open the **Domain Plot Parameters** dialog box, then go to the **General** page. In the **Plot type** area click the **Point plot** option button. Also select the **Keep current plot** check box at the bottom of the dialog box.

2 Click the **Title/Axis** button at the bottom of the dialog box. Near the **First axis label** text, click the option button next to the edit field and in that field type `Time [s]`. Click the option button for the **Second axis label** edit field and type `log(Q_tot)[J]`. Click **OK**.

3 Go to the **Point** page, then select Point 1. In the **Expression** edit field type `log10(uP+1)`. Click **Apply**, and a new figure appears.

4 In the **Expression** edit field, type `log10(uD+1)`.

5 Click the **Line Settings** button. In the **Line style** list, select **Dashed line**, then click **OK**.

6 Click **OK** to close the **Domain Plot Parameters** dialog box.

Convection Cooking of Chicken Patties

Introduction

This example models the convection cooking of a chicken patty. The model was originally developed by H. Chen and others (Ref. 1).

To increase consumer convenience, many of today's food products are precooked so that you can quickly re-heat the product, for example in a microwave oven. One industrial precooking method is air-convection cooking. This example builds a time-dependent model of the convection cooking process for a chicken patty, and it shows the temperature rise over time in the patty.

This simulation also models the moisture concentration in the patty, which is defined as the mass of water per volume of meat. From the viewpoint of product quality, it is of interest to minimize the loss of moisture during cooking. In this regard, cooking yield is a quantity that measures how much moisture, in percent, remains in the patty after the cooking process. Furthermore, the moisture concentration also influences the temperature field by heat loss due to vaporization and also by changing the patty's thermal conductivity.

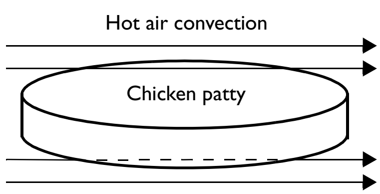


Figure 3-5: Convection cooking of a chicken patty.

Model Definition

This COMSOL Multiphysics example couples two time-dependent application modes describing the temperature and the moisture concentration, respectively. The simulation does not model the convective velocity field outside the patty because the coefficients for convective heat and moisture transfer to the surrounding air are given.

Inside the patty, diffusive processes describe both heat transfer and moisture transport. For the temperature, the heat equation describes the diffusive process as in

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot (-k \nabla T) = 0$$

where ρ is the patty's density (kg/m^3), C_p denotes the specific heat capacity (J/(kg·K)), T is the temperature (K), and k is the thermal conductivity (W/(m·K)). This model assumes that the specific heat capacity increases with temperature according to the expression

$$C_p = 3017.2 + 2.05\Delta T + 0.24(\Delta T)^2 + 0.002(\Delta T)^3 \quad (\mathrm{J/(kg{\cdot}K)})$$

where $\Delta T = (T - 0\ °\mathrm{C})$ and the dimensions of the numerical coefficients are such that the dimension of C_p is as stated.

For the moisture concentration, apply the diffusion equation

$$\frac{\partial c}{\partial t} + \nabla \cdot (-D \nabla c) = 0$$

where c is the moisture concentration (kg/m^3), and D is the diffusion coefficient (m^2/s).

Figure 3-6 depicts the patty's geometry, which is simple and allows for 2D axisymmetric modeling of its cross section. Additional symmetry in the cross section makes it possible to model just one quarter of the cross section.

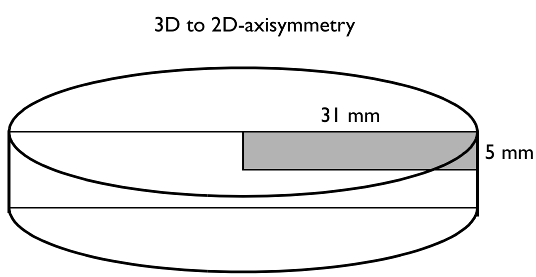


Figure 3-6: Geometry of the chicken patty.

These simplifications result in a simple rectangular domain with the dimension 31 mm × 5 mm. Figure 3-7 describes the boundary numbering used when specifying the boundary conditions.

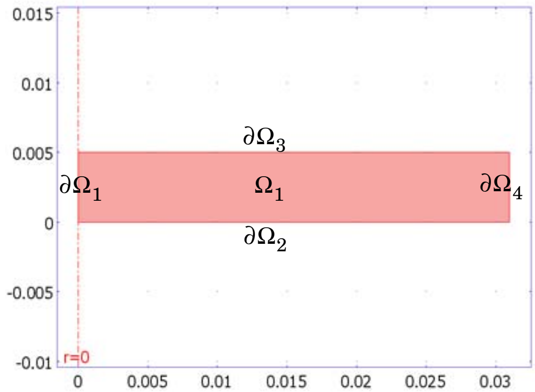


Figure 3-7: Model domain and boundary numbering.

The equations describing moisture diffusion are coupled to the heat equation in the following two ways:

- The thermal conductivity, k, increases with moisture concentration according to $k = (0.194 + 0.436(c/\rho))$ W/(m·K), where the concentration, c, and the density, ρ, must be expressed in the previously stated units.
- The vaporization of water at the patty's outer boundaries generates a heat flux out of the patty. Represent this heat flux with the term $D_m \lambda \nabla c$ in the boundary conditions for Boundaries 3 and 4, where D_m is the moisture diffusion coefficient (m^2/s) from the patty to the surrounding air and λ is the latent heat of vaporization (J/kg).

Assume symmetry for the temperature field on Boundaries 1 and 2. Air convection adds heat on Boundaries 3 and 4. According to the assumptions made earlier, add a term for the heat flux out of the patty due to moisture vaporization on Boundaries 3 and 4.

Summarizing, the boundary conditions for the general heat transfer application mode are

$$\mathbf{n} \cdot (-k \nabla T) = 0 \qquad \text{at } \partial\Omega_1 \text{ and } \partial\Omega_2$$
$$\mathbf{n} \cdot (k \nabla T) = h_T(T_{inf} - T) + \mathbf{n} \cdot (D_m \lambda \nabla c) \qquad \text{at } \partial\Omega_3 \text{ and } \partial\Omega_4$$

where h_T is the heat transfer coefficient (W/(m^2·K)), and T_{inf} is the oven air temperature.

The boundary conditions for the diffusion application mode are

$$\mathbf{n} \cdot (-D\nabla c) = 0 \qquad \text{at } \partial\Omega_1 \text{ and } \partial\Omega_2$$
$$\mathbf{n} \cdot (D\nabla c) = k_c(c_b - c) \qquad \text{at } \partial\Omega_3 \text{ and } \partial\Omega_4$$

where D is the moisture diffusion coefficient in the patty (m^2/s), k_c refers to the mass transfer coefficient (m/s), and c_b denotes the outside air (bulk) moisture concentration (kg/m^3). The diffusion coefficient and the mass transfer coefficient are given, respectively, by

$$D = \frac{k_m}{\rho C_m}, \qquad k_c = \frac{h_m}{\rho C_m},$$

where C_m equals the specific moisture capacity (kg moisture/kg meat), k_m refers to the moisture conductivity (kg/(m·s)), and h_m denotes the mass transfer coefficient in mass units ($kg/(m^2 \cdot s)$).

Assume that the patty's temperature is 22 °C at the start of the cooking process, and the moisture concentration of the air is 22 kg/m^3 on a wet basis, which means that the moisture is expressed in mass per volume of meat. Additional data are given in the modeling section below.

To obtain the temperature and moisture concentration over time, the model solves the equations with the boundary conditions discussed above.

Results and Discussion

The most interesting result from this simulation is the time required to heat the patty from room temperature (22 °C) to at least 70 °C throughout the entire patty. The section at the middle of the patty (at the lower left corner of the modeling domain) takes the longest time to reach this temperature. It is also interesting to determine how much moisture remains in the patty after cooking. For this purpose, compute the cooking yield, defined as (initial moisture mass)/(final moisture mass).

The model shows that at an oven air temperature of 135 °C, a cooking time of 840 s is required to reach a center temperature of 70 °C. Figure 3-8 shows how the temperature increases over time.

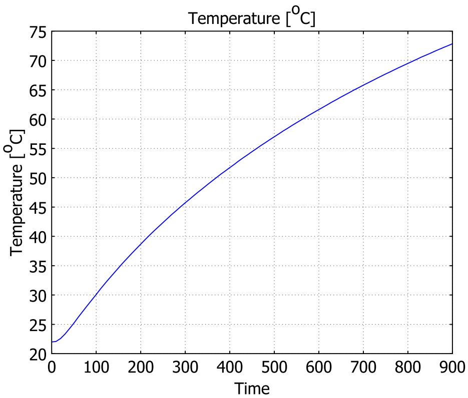


Figure 3-8: Temperature increase over time in the middle of the patty at an air temperature of 135 °C.

Figure 3-9 illustrates the resulting temperature field after 840 s. The temperature at the lower left corner is 70 °C, and the temperature rises toward the outside boundaries.

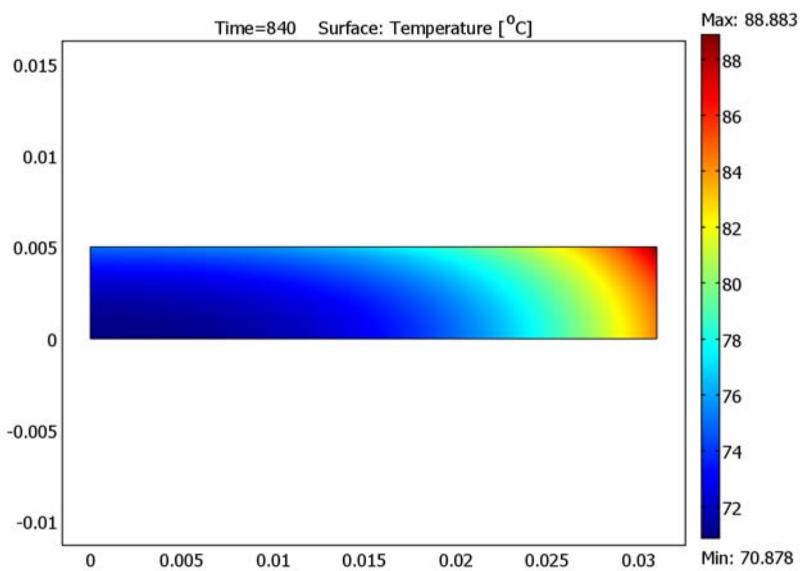


Figure 3-9: Temperature field after 840 s at a cooking temperature of 135 °C.

At this oven air temperature, the cooking yield is approximately 0.93 (93%). Figure 3-10 shows the resulting moisture concentration for these conditions. As expected, note that the convective loss of moisture at the boundaries results in a lower moisture concentration at the outer parts of the patty compared to its inner parts.

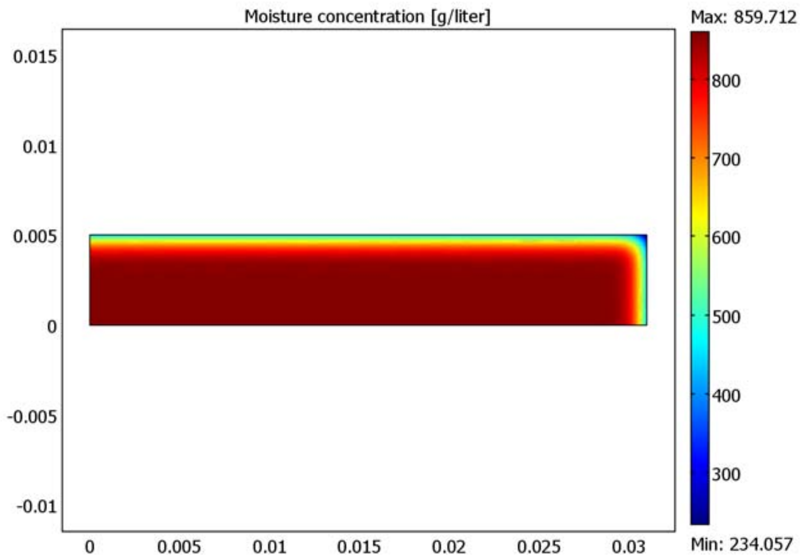


Figure 3-10: Moisture concentration after 840 s at a cooking temperature of 135 °C.

Simulations show that an increased air temperature both shortens the time required to reach 70 °C in the middle and increases the cooking yield. The drawback, however, is that the temperature gradients in the chicken patty increase. Figure 3-11 shows the temperature field obtained after 370 s at a cooking temperature of 218 °C; the corresponding cooking yield is 0.97 (97%).

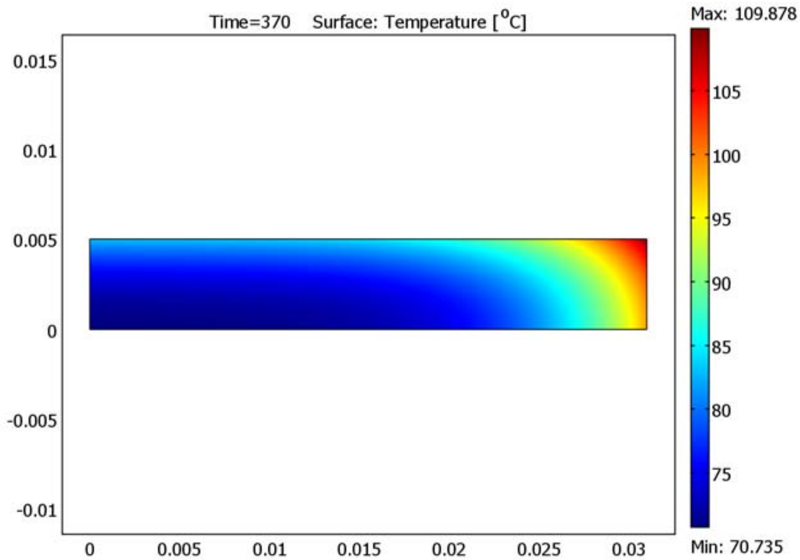


Figure 3-11: Temperature field after 370 s at a cooking temperature of 218 °C.

Reference

1. H. Chen, B.P. Marks and R.Y. Murphy, "Modeling coupled heat and mass transfer for convection cooking of chicken patties," *Journal of Food Engineering*, vol. 42, 1999, pp. 139–146.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/chicken_patties`

MODEL NAVIGATOR

1. Open the **Model Navigator** and go to the **New** page. From the **Space dimension** list select **Axial symmetry (2D)**.
2. From the list of application modes select **Heat Transfer Module>General Heat Transfer>Transient analysis**.
3. Click the **Multiphysics** button, then click **Add**.
4. Similarly select the application mode **COMSOL Multiphysics> Convection and Diffusion>Diffusion>Transient analysis**, then click **Add**.
5. Click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu select **Constants**. Enter the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_air	135[degC]	Oven air temperature
T0	22[degC]	Initial patty temperature
rho	1100[kg/m^3]	Density of patty
h_T	25[W/(m^2*K)]	Heat transfer coefficient
c0	0.78*rho	Initial moisture concentration
c_b	0.02*rho	Air moisture concentration
C_m	0.003	Specific moisture capacity
k_m	1.29e-9[kg/(m*s)]	Moisture conductivity
h_m	1.67e-6[kg/(m^2*s)]	Mass transfer coefficient in mass units
D	k_m/(rho*C_m)	Diffusion coefficient
k_c	h_m/(rho*C_m)	Mass transfer coefficient
D_m	5e-10[m^2/s]	Surface moisture diffusivity
lda	2.3e6[J/kg]	Latent heat of vaporization

GEOMETRY MODELING

1. Press the Shift key and click the **Rectangle/Square** button on the Draw toolbar.

2 In the dialog box that appears, enter the rectangle properties given below; when done, click **OK**.

OBJECT DIMENSIONS	EXPRESSION
Width	31e-3
Height	5e-3
Base	Corner
Position, r	0
Position, z	0

3 Click the **Zoom Extents** button on the Main toolbar.

PHYSICS SETTINGS

From the **Options** menu select **Expressions>Subdomain Expressions**. In the dialog box enter the following details; when done, click **OK**.

NAME	EXPRESSION
k_T	(0.194+0.436[kg/mol]*c/rho)[W/(m*K)]
dT	(T-0[degC])[1/K]
C_p	(3017.2+2.05*dT+0.24*dT^2+0.002*dT^3)[J/(kg*K)]

The unit label "[kg/mol]" is inserted in the expression for k_T because the dependent variable in the Diffusion application mode, the concentration c, has the default unit mol/m^3; the above insertion gives k_T the correct dimension. In this model, whenever the concentration unit mol/m^3 appears in the user interface, read instead kg/m^3.

Boundary Conditions—General Heat Transfer

1 From the **Multiphysics** menu select **1 General Heat Transfer (htgh)**.

2 From the **Physics** menu select **Boundary Settings**.

3 Enter the settings from the following table; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARY 2	BOUNDARY 3	BOUNDARY 4
Boundary condition	Axial symmetry	Insulation/ Symmetry	Heat flux	Heat flux
q_0			`D_m*lda*cz`	`D_m*lda*cr`
h			`h_T`	`h_T`
T_{inf}			`T_air`	`T_air`

Following standard COMSOL Multiphysics syntax, the variables `cz` and `cr` represent the concentration-gradient components $\partial c/\partial z$ and $\partial c/\partial r$, respectively.

Note: If the preference **Highlight unexpected units** is set (on the **Modeling** page of the **Preferences** dialog box that you open from the **Options** menu), the entries in the $\mathbf{q_0}$ edit field for Boundaries 3 and 4 appear in red. This is because, as just mentioned, the software expects concentrations to be given in amount of substance per unit volume (with SI unit mol/m^3). Because, in this model, the concentration is consistently expressed in mass per unit volume, you can just ignore this warning.

Subdomain Settings—General Heat Transfer

1 From the **Physics** menu select **Subdomain Settings**.

2 Select Subdomain 1, and note that only conductive heat transfer is enabled by default.

3 Click the **General** tab, then enter properties for the chicken meat as in the following table:

SETTINGS	SUBDOMAIN 1
k (isotropic)	`k_T`
ρ	`rho`
C_p	`C_p`

4 Click the **Init** tab. In the **Temperature** edit field type `T0`, then click **OK**.

Boundary Conditions—Diffusion

1 From the **Multiphysics** menu select **2 Diffusion (di)**.

2 From the **Physics** menu open the **Boundary Settings** dialog box. Enter boundary coefficients as in the following table; when done, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARY 2	BOUNDARIES 3, 4
Boundary condition	Axial symmetry	Insulation/ Symmetry	Flux
k_c			k_c
c_b			c_b

Subdomain Settings—Diffusion

1 From the **Physics** menu open the **Subdomain Settings** dialog box. In the **Subdomain selection** list select **1**, then go to the **D isotropic** edit field and type `D`.

2 Go to the **Init** page, then in the **c(t_0)** edit field type `c0`.

3 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu select **Free Mesh Parameters**.

2 Click the **Boundary** tab. In the **Boundary selection** list choose **3** and **4**. In the **Maximum element size** edit field type `1e-3`.

3 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

1 From the **Solve** menu open the **Solver Parameters** dialog box.

2 On the **General** page find the **Time stepping** area. In the **Times** edit field type `0:10:900`, then click **OK**.

3 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the temperature in the chicken patty (in kelvin) at $t = 900$ s. To generate Figure 3-9, follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page, from **Solution at time** list select **840**.

3 Click the **Surface** tab. On the **Surface Data** page, select **°C** from the **Unit** list.

4 Click **Apply**.

To generate Figure 3-10, which shows the moisture concentration, proceed as follows:

1 While still on the **Surface** page, in the **Expression** edit field type `c*1[kg/mol]`, then press Enter.

The entry in the **Unit** list should now read **kg/m^3**, which is the same as g/liter.

2 Click the **General** tab, then click the **Title** button.

3 In the **Title** dialog box, select the option button next to the edit field, then enter the title `Moisture concentration [g/liter]`. Click **OK**.

4 Click **OK** to generate the plot.

To create Figure 3-8, which shows temperature versus time, follow these instructions:

1 From the **Postprocessing** menu select **Domain Plot Parameters**.

2 On the **General** page, find the **Plot type** area and select the **Point plot** option button.

3 Click the **Point** tab. In the **Point selection** list select **1** and in the **Unit** list select **°C**.

4 Click **OK**.

The following steps describe how to compute the cooking yield:

1 From the **Postprocessing** menu open the **Subdomain Integration** dialog box.

2 Make sure the **Compute volume integral** check box is selected.

3 Select Subdomain 1, then in the **Expression** edit field type `(c[kg/mol]/c0)/1.509e-5[m^3]`. The denominator, $1.509 \cdot 10^{-5}\ \mathrm{m}^3$, is the value of the volume integral of the modeling geometry.

4 Click **OK** to obtain the cooking yield; the result (approximately 0.93) appears in the message log at the bottom of the user interface.

To investigate the model further, you can solve the problem for other air temperatures using the same steps for postprocessing.

Cooling Flange

Introduction

In the chemical industry, processes often cool reaction fluids using glass flanges. In most cases the coolant is the surrounding air. An obvious design parameter for this type of device is the cooling power, and the surface temperatures might also be of interest. Heat transfer in this type of device is dominated by convection to and from the surfaces, although the conduction within the glass flange can also influence performance. A convenient method to analyze convection cooling is to use a heat transfer coefficient, h. This coefficient describes the influence of the fluid-flow field and the convective fluxes. Thus it is not necessary to model the flow field, which greatly simplifies simulations.

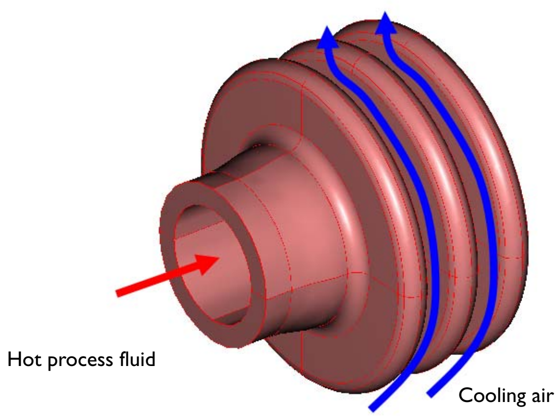


Figure 3-12: Operating principle of the cooling flange.

Semi-empirical expressions for computing the heat transfer coefficient for different cases are available in the literature. For this model, the author obtained the heat transfer coefficient for the outer surface by using semi-empirical data available for natural convection around a cylinder. The heat transfer coefficient for the surface that faces the tube is valid for forced convection in a tube. The model uses the General Heat Transfer application mode.

Model Definition

Figure 3-13 presents a drawing of the modeled geometry.

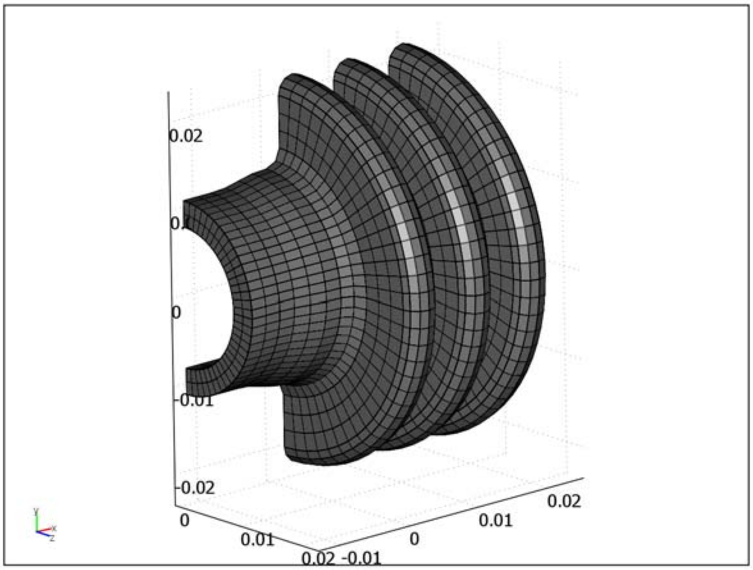


Figure 3-13: Drawing of the cooling flange.

The pipe connecting the flange has an inner diameter of 16 mm and a wall thickness of 3 mm. In the flange section, the pipe is 4 mm thick. The flanges are 4 mm thick and 10 mm in height.

During operation, the hot process fluid heats the inside of the tube. The flange conducts the heat and transfers it to the surrounding air. As the air is heated, buoyancy effects cause a convective flow.

The heat transfer within the flange is described by the stationary heat equation

$$\nabla \cdot (-k\nabla T) = 0$$

where k is the thermal conductivity (W/(m·K)), and T is the temperature (K). On the flange's exterior boundaries, which face the air and process fluid, the applicable boundary condition is

$$-\mathbf{n} \cdot (-k\nabla T) = q_0 + h(T_{\mathrm{inf}} - T)$$

where $\mathbf{n}$ is the normal vector of the boundary, h is the heat transfer coefficient (W/(m^2·K)), and T_{inf} is the temperature of the surrounding medium (K). For this simulation, set T_{inf} to 298 K for the cooling air and to 363 K for the process fluid.

You can approximate the value for the heat transfer coefficient, h, on the process fluid side with a constant value of 15 W/(m^2·K)) because the fluid's velocity is close to constant and the model assumes that its temperature decreases only slightly.

The h expression on the air side is more elaborate. Assume that the free-convection process around the flange is similar to that around a cylinder. The heat transfer coefficient for a cylinder is available in the literature (Ref. 1), and you can use the expression:

$$h = \frac{k}{L} f(\theta)\,\mathrm{Gr}^{1/4}$$

where k is the thermal conductivity of air (0.06 W/(m·K)), L is the characteristic length that in this case is the outer diameter of the flange (44 mm), and $f(\theta)$ is an empirical coefficient tabulated in Table 3-2. (Figure 3-14 illustrates the definition of the angle θ). Finally, Gr is the Grashof number defined as

$$\mathrm{Gr} = \frac{\beta g \Delta T L^3}{\mu^2}$$

where β is the thermal expansion coefficient (1/K), which equals $1/T_\infty$ for an ideal gas, g is the gravitational acceleration (9.81 m/s^2), and μ is the kinematic viscosity ($18 \cdot 10^{-6}$ Pa·s).

TABLE 3-2: EMPIRICAL TRANSFER COEFFICIENTS

INCIDENT ANGLE [DEG.]	$f(\theta)$
0	0.48
90	0.46
100	0.45
110	0.435
120	0.42
130	0.38
140	0.35
150	0.28
160	0.22
180	0.15

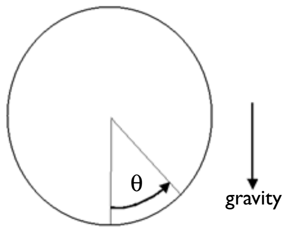


Figure 3-14: Definition of the angle θ.

Table 3-3 summarizes the material properties of the flange material.

TABLE 3-3: MATERIAL PROPERTIES

MATERIAL	k [W/(m·K)]	ρ [kg/m^3]	C_p [J/(kg·K)]
Silica glass	1.38	2203	703

Results and Discussion

Figure 3-16 shows the surface temperature of the flange at steady state.

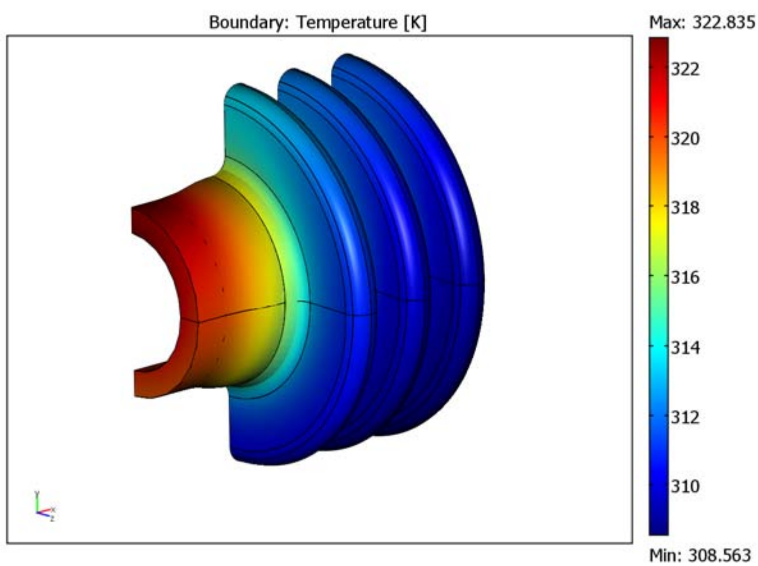


Figure 3-15: Stationary surface temperature of the flange.

The temperature at the flange shoulders is approximately 14 K lower than that at the tube surface. The temperature difference between the process fluid at 363 K and the inner surface of the pipe is approximately 40 K, while that between the outer flange surface and the air stream is approximately 10 K. These values indicate that the heat transfer from the flange outer surfaces is efficient. It also indicates that the heat transfer from the fluid to the flange is a limiting factor. To improve the flange's performance, it is a good idea to increase the tube diameter.

Figure 3-16 shows the heat transfer coefficient for the flange and pipe walls.

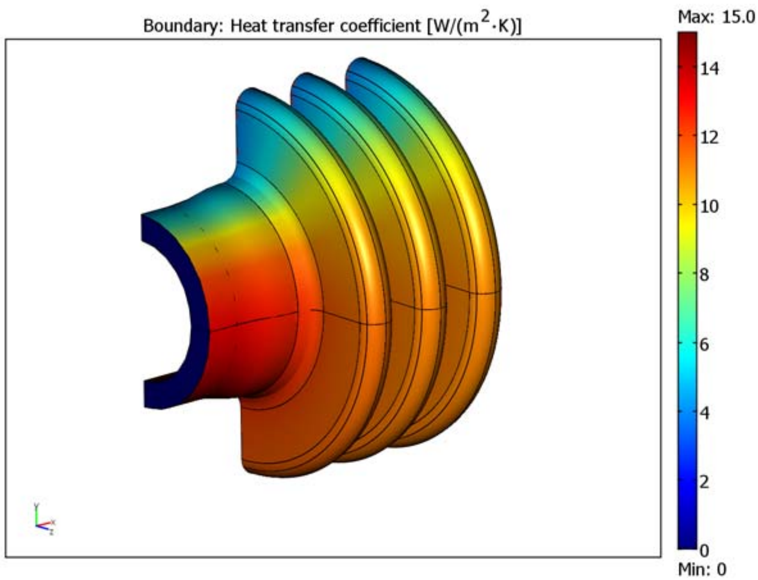


Figure 3-16: Heat transfer film coefficient, h, for the flange.

As you can see, the coefficient decreases significantly along the vertical position of the flange's outer boundary.

Calculate the flange's total cooling power by integrating the heat flux on the outer surfaces. The entire flange, that is, taking both symmetry halves into account, has a cooling power of approximately 1.2 W.

Reference

1. B. Sundén, *Kompendium i värmeöverföring [Notes on Heat Transfer]*, Sec. 10-3, Dept. of Heat and Power Engineering, Lund Inst. of Technology, 2003 (in Swedish).

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/cooling_flange`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

Open the **Model Navigator** and go to the **New** page. In the **Space dimension** list select **2D**, then click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu select **Constants**. Define the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
D	44[mm]	Outer flange diameter
k	0.06[W/(m*K)]	Thermal conductivity
Tair	298[K]	Cooling air temperature
Tinner	363[K]	Process fluid temperature
Hh	15[W/(m^2*K)]	Heat transfer coefficient
visc	18e-6[m^2/s]	Kinematic viscosity
beta	1/Tair	Thermal expansion coefficient
grav	9.81[m/s^2]	Gravitational acceleration

2 From the **Options** menu select **Functions**. In the dialog box that appears, click **New** to open the **New Function** dialog box.

3 In the **Function name** edit field, type `graph`. Click **Interpolation** and select **Table** in the **Use data from** list. Click **OK**.

4 Select **Linear** in the **Interpolation method** list in the **Functions** dialog box that appears. Enter the following values in the columns for **x** and **f(x)**; when done, click **OK**.

x	f(x)
0	0.48
90	0.46
100	0.45
110	0.435
120	0.42
130	0.38
140	0.35
150	0.28
160	0.22
180	0.15

5 From the **Options** menu select **Axes/Grid Settings**.

6 Clear the **Axis equal** check box, then enter the properties in the following table:

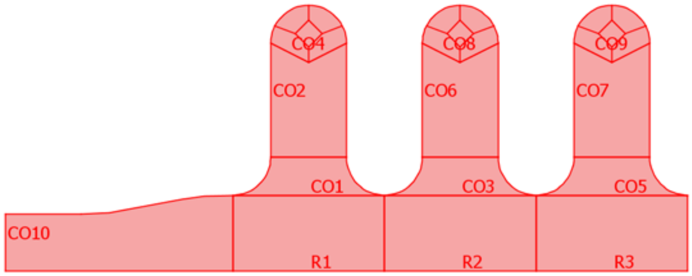
PROPERTY	VALUE
x min	-1.4
x max	2.6
y min	0.2
y max	2.8

7 Go to the **Grid** page. Clear the **Auto** check box, then enter the properties in the following table; when done, click **OK**.

PROPERTY	VALUE
x spacing	0.1
y spacing	0.1

GEOMETRY MODELING

Start by creating this 2D geometry:



Create the rectangle R1 with these steps:

1. Shift-click the **Rectangle/Square** button on the Draw toolbar.
2. In the dialog box that appears, enter the following settings; when done, click **OK**.

OBJECT DIMENSIONS	EXPRESSION
Width	0.8
Height	0.4
Base	Corner
x-position	0
y-position	0.8

Create the composite object CO1 with these steps:

1. Click the **2nd Degree Bézier Curve** button on the Draw toolbar.
2. Click on the coordinates (0, 1.2), (0.2, 1.2), and (0.2, 1.4).
3. Click the **Line** button on the Draw toolbar, then click on the coordinate (0.6, 1.4).
4. Click the **2nd Degree Bézier Curve** on the Draw toolbar.
5. Draw the Bézier curve by clicking on the coordinates (0.6, 1.4), (0.6, 1.2), and (0.8, 1.2).
6. Click the right mouse button to create the geometry object.

Create the composite object CO2 with these steps:

1. Click the **Line** button on the Draw toolbar, then click on the coordinates (0.2, 1.4), (0.2, 2.0), (0.4, 1.9), (0.6, 2.0), and (0.6, 1.4).
2. Click the right mouse button to create the geometry object.

Create the composite object CO4 with these steps:

1 Click the **2nd Degree Bézier Curve** button on the Draw toolbar.

2 Click on the coordinates (0.2, 2.0), (0.2, 2.2), (0.4, 2.2), (0.6, 2.2), and (0.6, 2.0).

3 Click the **Line** button on the Draw toolbar, then click on the coordinates (0.4, 1.9) and (0.2, 2.0). Click the right mouse button.

   This creates a composite geometry object, CO3. To create CO4, you still must add a square and some lines to the object.

4 Shift-click the **Rectangle/Square (Centered)** button on the Draw toolbar.

5 In the dialog box that appears, enter the following settings; when done, click **OK**.

PROPERTY	VALUE
Width	0.1
α	45
Base	Center
x-position	0.4
y-position	2.05

6 Click the **Line** button on the Draw toolbar. Click on the left corner of the small rectangle, then click on the coordinate (0.2, 2.1). Click the right mouse button.

7 Click the **Line** button on the Draw toolbar, click on the right corner of the small rectangle, then click on the coordinate (0.6, 2.1). Click the right mouse button.

8 Click the **Line** button on the Draw toolbar, click on the upper corner of the small rectangle, then click on the coordinate (0.4, 2.2). Click the right mouse button.

9 Click the **Line** button on the Draw toolbar, click on the lower corner of the small rectangle, then click on the coordinate (0.4, 1.9). Click the right mouse button.

10 To create the composite object, select CO3, SQ1, B1, B2, B3, and B4, then click the **Coerce to Solid** button on the Draw toolbar.

Create the other two flanges with these steps:

1 Select R1, CO1, CO2, and CO4, then click the **Array** button on the Draw toolbar.

2 In the dialog box that appears, enter the following properties; when done, click **OK**.

PROPERTY	VALUE
Displacement, x	0.8
Displacement, y	0

PROPERTY	VALUE
Array size, x	3
Array size, y	1

Create the composite object CO10 with these steps:

1 Click the **Line** button on the Draw toolbar.

2 Click on the lower left corner of rectangle R1, then click, in order, on the coordinates (–1.2, 0.8), (–1.2, 1.1), and (–0.8, 1.1).

3 Click the **3rd Degree Bézier Curve** button on the Draw toolbar.

4 Click, in order, on the coordinates (–0.4, 1.1), (–0.4, 1.2), and (0, 1.2).

5 Click the right mouse button to create the geometry object CO10.

Conclude the geometry modeling by scaling the geometry with these steps:

1 Select all geometry objects by pressing Ctrl+A.

2 Click the **Scale** button on the Draw toolbar.

3 In the dialog box that appears, enter the following scaling properties; when done, click **OK**.

PROPERTY	VALUE
Scale factor, x	`0.01`
Scale factor, y	`0.01`
Scale base point, x	0
Scale base point, y	0

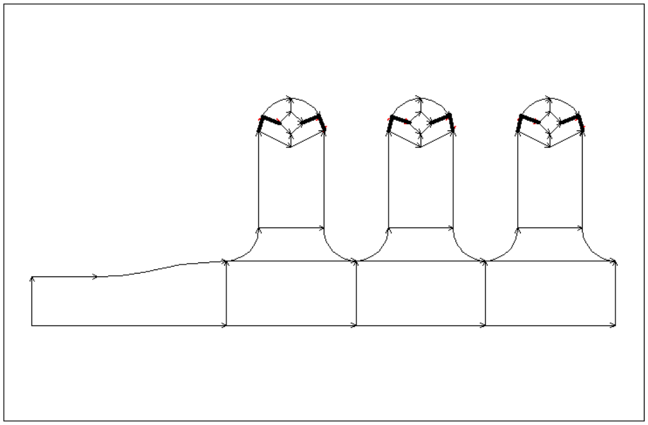
4 Double-click the **EQUAL** button on the status bar at the bottom of the user interface.

5 Click the **Zoom Extents** button on the Main toolbar.

MESH GENERATION

1 From the **Mesh** menu select **Mapped Mesh Parameters**.

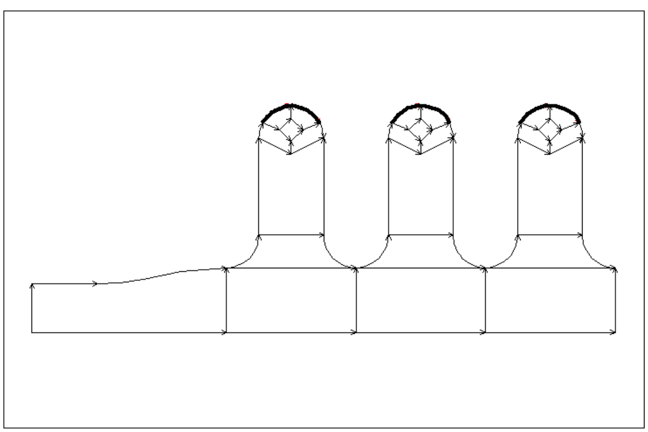
2 On the **Boundary** page, select the boundaries marked in the following figure:



.

3 Select the **Constrained edge element distribution** check box (keep the default value for the **Number of edge elements** at 1).

4 Select the boundaries marked in the following figure:



5 Select the **Constrained edge element distribution** check box, then in the **Number of edge elements** edit field type 2.

6 Click **Remesh**, then click **OK**.

7 From the **Mesh** menu select **Revolve Mesh**.

8 In the dialog box that appears, in the α**2** edit field type 180.

9 Click the **Angle from x-axis** button, and in the θ edit field type 0.

10 Click **OK**.

PHYSICS SETTINGS

1 From the **Multiphysics** menu select **Model Navigator**.

2 In the **Multiphysics** menu on the right side of the dialog box select **Geom2 (3D)**.

3 From the list of application modes on the left side of the dialog box select **Heat Transfer Module>General Heat Transfer>Steady-state analysis**.

4 Click **Add**, then click **OK**.

Subdomain Settings

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select all subdomains.

Next, select the material properties from the materials library:

3 Press the **Load** button. In the dialog box that appears, in the **Materials** list select **Library1>Silica glass**. Click **OK**.

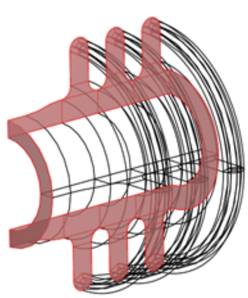
4 Click the **Init** tab, then in the **Temperature** edit field type 323.

5 Click **OK**.

Boundary Settings

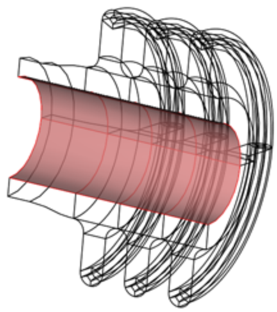
1 From the **Physics** menu select **Boundary Settings**.

2 Select the boundaries at the geometry's ends and the symmetry boundaries, as shown in the following figure:



3 In the **Boundary condition** list select **Insulation/Symmetry**.

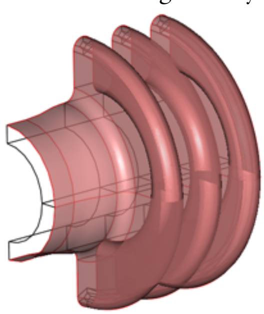
4 Select the boundaries that face the inner channel, as shown in the following figure:



5 In the **Boundary condition** list select **Heat flux**, then enter the following properties:

PROPERTY	VALUE
h	`Hh`
T_{inf}	`Tinner`

6 Select Boundary 3, then select the **Select by group** check box. All boundaries on the outside of the geometry should now be selected, as in the following figure:



7 In the **Boundary condition** list select **Heat flux**, then enter the following properties:

PROPERTY	VALUE
h	`Hc`
T_{inf}	`Tair`

8 Click **OK**.

9 From the **Options** menu select **Expressions>Boundary Expressions**. Keep the selection of all outward-facing boundary segments from Step 6.

10 Enter the following expressions; when done, click **OK**.

EXPRESSION	VALUE
`angle`	`atan(y/z)*180/pi+90`
`Gr`	`grav*beta*(T-Tair)*D^3/visc^2`
`Hc`	`k*graph(angle)*Gr^0.25/D`

COMPUTING THE SOLUTION

1. Click the **Solver Parameters** button on the Main toolbar.
2. In the **Linear system solver** list select **GMRES**.
3. In the **Preconditioner** list select **Algebraic multigrid**.
4. Click **OK**.
5. Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To reproduce the plot in Figure 3-15, displaying the boundary temperature distribution, follow these instructions:

1. Click the **Plot Parameters** button on the Main toolbar.
2. Clear the **Slice** check box and select the **Boundary** check box, then click **Apply**.
3. Double-click the **GRID** button on the status bar at the bottom of the user interface.

To reproduce the plot in Figure 3-16, continue with these steps:

1. Still in the **Plot Parameters** dialog box, click the **Boundary** tab.
2. In the **Predefined quantities** list select **Heat transfer coefficient**, then click **OK**.

To integrate the heat flux over the outer surface area of the flange, do as follows:

1. From the **Postprocessing** menu open the **Boundary Integration** dialog box.
2. In order to select all outer boundaries, open the **Boundary Settings** dialog box, then select Boundary 3 and select the **Select by group** check box. Click **Cancel** to close the **Boundary Settings** dialog box.
3. Return to the **Boundary Integration** dialog box. From the **Predefined quantities** list select **Normal total heat flux**, then click **OK**.

The integrated value appears in the message log at the bottom of the user interface.

Friction Stir Welding

Introduction

Manufacturers use a modern welding method called friction stir welding to join aluminum plates. This model analyzes the heat transfer in this welding process. The model is based on a paper by M. Song and R. Kovacevic (Ref. 1).

In friction stir welding, a rotating tool moves along the weld joint and melts the aluminum through the generation of friction heat. The tool's rotation stirs the melted aluminum such that the two plates are joined. Figure 3-17 shows the rotating tool and the aluminum plates being are joined.

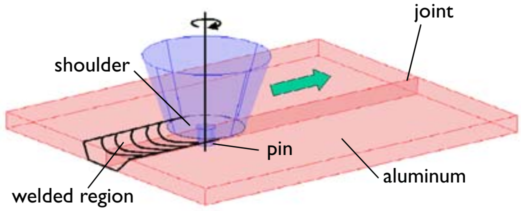


Figure 3-17: Two aluminum plates being joined by friction stir welding.

The rotating tool is in contact with the aluminum plates along two surfaces: the tool's *shoulder*, and the tool's *pin*. The tool adds heat to the aluminum plates through both interfaces.

During the welding process, the tool moves along the weld joint. This movement would require a fairly complex model if you want to model the tool as a moving heat source. This example takes a different approach that uses a moving coordinate system that is fixed at the tool axis (Ref. 1 also takes this approach). After making the coordinate transformation, the heat transfer problem becomes a stationary convection-conduction problem that is straightforward to model.

The model includes some simplifications. For example, the coordinate transformation assumes that the aluminum plates are infinitely long. This means that the analysis neglects effects near the edges of the plates. Neither does the model account for the stirring process in the aluminum, which is very complex because it includes phase changes and material flow from the front to the back of the rotating tool.

Model Definition

The model geometry is symmetric around the weld. It is therefore sufficient to model only one aluminum plate. The plate dimensions are 120 mm × 102 mm × 12.7 mm, surrounded by two infinite domains in the x direction. Figure 3-18 shows the resulting model geometry:

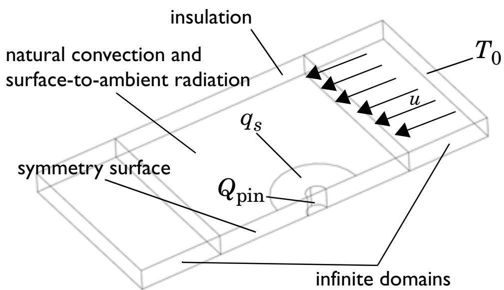


Figure 3-18: Model geometry for friction stir welding.

The following equation describes heat transfer in the plate. As a result of fixing the coordinate system in the welding tool, the equation includes a convective term in addition to the conductive term. The equation is

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

where k represents thermal conductivity, ρ is the density, C_p denotes specific heat capacity, and $\mathbf{u}$ is the velocity.

The model sets the velocity to $u = 1.59 \cdot 10^{-3}$ m/s in the negative x direction.

The model simulates the heat generated in the interface between the tool's pin and the workpiece as a surface heat source (expression adapted from Ref. 2):

$$q_{\mathrm{pin}}(T) = \frac{\mu}{\sqrt{3(1+\mu^2)}} r_{\mathrm{p}} \omega \bar{Y}(T) \quad (\mathrm{W/m^2})$$

Here μ is the friction coefficient, r_{p} denotes the pin radius, ω refers to the pin's angular velocity (rad/s), and $\bar{Y}(T)$ is the average shear stress of the material. As indicated, the average shear stress is a function of the temperature; for this model, you approximate this function with an interpolation function determined from experimental data given in Ref. 1 (see Figure 3-20).

Additionally, heat is generated at the interface between the tool's shoulder and the workpiece; the following expression defines the local heat flux per unit area (W/m^2) at the distance r from the center axis of the tool:

$$q_{\mathrm{shoulder}}(r, T) = \begin{cases} (\mu F_{\mathrm{n}}/A_s)\omega r & ; T < T_{\mathrm{melt}} \\ 0 & ; T \geq T_{\mathrm{melt}} \end{cases}$$

Here F_n represents the normal force, A_{s} is the shoulder's surface area, and T_{melt} is aluminum's melting temperature. As before, μ is the friction coefficient and ω is the angular velocity of the tool (rad/s).

Above the melting temperature of aluminum, the friction between the tool and the aluminum plate is very low. Therefore, the model sets the heat generation from the shoulder and the pin to zero when the temperature is equal to or higher than the melting temperature.

Symmetry will be assumed along the weld joint boundary.

The upper and lower surfaces of the aluminum plates lose heat due to natural convection and surface-to-ambient radiation. The corresponding heat flux expressions for these surfaces are

$$q_{\mathrm{up}} = h_{\mathrm{up}}(T_0 - T) + \varepsilon\sigma(T_{\mathrm{amb}}^4 - T^4)$$
$$q_{\mathrm{down}} = h_{\mathrm{down}}(T_0 - T) + \varepsilon\sigma(T_{\mathrm{amb}}^4 - T^4)$$

where h_{up} and h_{down} are heat transfer coefficients for natural convection, T_0 is an associated reference temperature, ε is the surface emissivity, σ is the Stefan-Boltzmann constant, and T_{amb} is the ambient air temperature.

The modeling of an infinite domain on the left-hand side, where the aluminum leaves the computational domain, makes sure that the temperature is in equilibrium with the temperature at infinity through natural convection and surface-to-ambient radiation. You therefore set the boundary condition to insulation at that location.

You can compute values for the heat transfer coefficients using empirical expressions available in the heat-transfer literature, for example, Ref. 3. In this model, use the values $h_{\mathrm{up}} = 12.25\ \mathrm{W/(m^2{\cdot}K)}$ and $h_{\mathrm{down}} = 6.25\ \mathrm{W/(m^2{\cdot}K)}$

Results and Discussion

Figure 3-19 shows the resulting temperature field. Consider this result as what you would see through a window fixed to the moving welding tool.

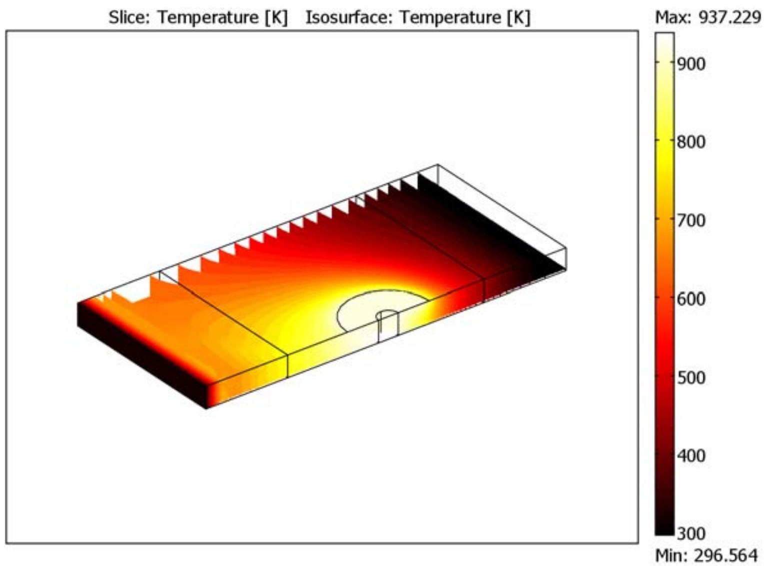


Figure 3-19: Temperature field in the aluminum plate.

The temperature is highest where the aluminum is in contact with the rotating tool. Behind the tool, the process transports hot material away, while in front of the tool, new cold material enters.

References

1. M. Song and R. Kovacevic, *International Journal of Machine Tools & Manufacture*, vol. 43, pp. 605–615, 2003.

2. P. Colegrove and others, "3-dimensional flow and thermal modelling of the friction stir welding process," Proceedings of the 2nd International Symposium on Friction Stir Welding, Gothenburg, Sweden, 2000.

3. A. Bejan, *Heat Transfer*, Wiley, 1993.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/friction_welding`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator** and click the **New** tab. In the **Space dimension** list select **3D**.

2 From the list of application modes select **Heat Transfer Module>General Heat Transfer>Steady-state analysis**.

3 Click **OK**.

OPTIONS AND SETTINGS

Constants

From the **Options** menu, select **Constants**. Define the following names, expressions, and (optionally) descriptions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T0	300[K]	Reference temperature
T_melt	933[K]	Workpiece melting temperature
rho_pin	7800[kg/m^3]	Pin density
k_pin	42[W/(m*K)]	Thermal conductivity
Cp_pin	500[J/(kg*K)]	Specific heat capacity
h_upside	12.25[W/(m^2*K)]	Heat transfer coefficient, upside
h_downside	6.25[W/(m^2*K)]	Heat transfer coefficient, downside
epsilon	0.3	Surface emissivity
u_weld	1.59[mm/s]	Welding speed
mu	0.4	Friction coefficient
n	637[1/min]	Rotation speed (RPM)
omega	2*pi[rad]*n	Angular velocity (rad/s)
F_n	25[kN]	Normal force
r_pin	6[mm]	Pin radius
r_shoulder	25[mm]	Shoulder radius
A_s	pi*(r_shoulder^2 -r_pin^2)	Shoulder surface area

Functions

Next, define an interpolation function for the aluminum yield stress, $\bar{Y}$, as a function of the temperature, T, based on experimental material data listed in Ref. 1.

1 From the **Options** menu open the **Functions** dialog box.

2 Click **New**.

3 In the **New Function** dialog box, type `Ybar` in the **Function name** edit field.

4 Select the **Interpolation** option button, then click **OK**.

5 Back in the **Functions** dialog box, leave the default settings for **Interpolation method** and **Extrapolation method**. In the **x** and **f(x)** columns enter the following data:

x	311	339	366	394	422	450	477	533	589	644
f(x)	241	238	232	223	189	138	92	34	19	12

The **x** values are temperatures (in kelvin) and the **f** values corresponding yield stresses (in MPa) for 6061-T6 aluminum.

6 When finished, click **Plot** to view the resulting interpolation function (Figure 3-20), then click **OK** to close the **Functions** dialog box.

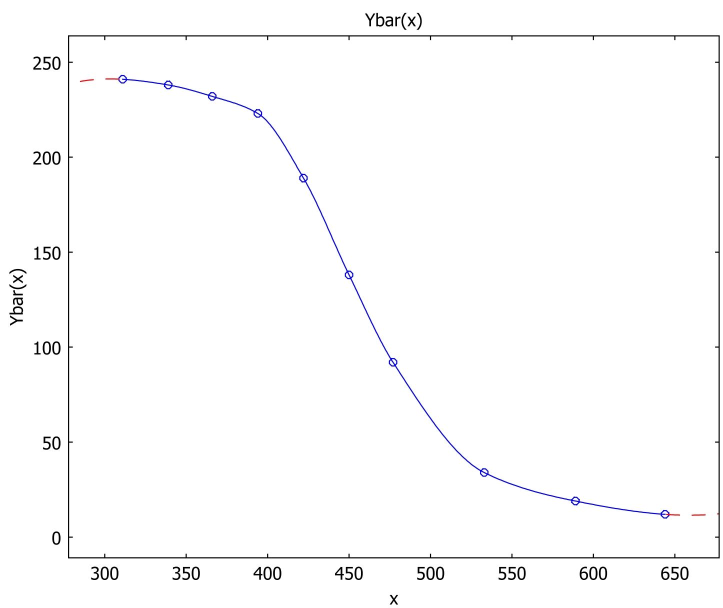


Figure 3-20: Yield stress (MPa) vs. temperature (K) for 6061-T6 aluminum.

GEOMETRY MODELING

1 Create the aluminum plate. From the **Draw** menu, select **Block**.
2 In the dialog box that appears, go to the **Length** area and enter the settings in the following table; when done, click **OK**.

OBJECT DIMENSIONS	EXPRESSION
X	`120e-3`
Y	`102e-3`
Z	`12.7e-3`

3 Repeat Steps 1 and 2 to draw another block using the settings in the following table:

OBJECT DIMENSIONS	EXPRESSION
X	`50e-3`
Y	`102e-3`
Z	`12.7e-3`

4 In the **Axis base point** area, type `-50e-3` in the **x** edit field. Click **OK**.
5 To create a third block of the same dimensions as the one you just created, press Ctrl+C and then press Ctrl+V. In the **Paste** dialog box that appears, type `170e-3` in the **x** edit field, then click **OK**.
6 Click the **Zoom Extents** button on the Main toolbar.
7 Go to the **Draw** menu and select **Work-Plane Settings**.
8 On the **Quick** page, click the **x-y** option button, then in the **z** edit field enter `12.7e-3`. Click **OK**.
9 Click the **Zoom Extents** button on the Main toolbar.
10 Press the Shift key and click the **Ellipse/Circle (Centered)** button on the Draw toolbar.
11 In the dialog box that appears, enter the following circle properties; when done, click **OK**:

OBJECT DIMENSIONS	EXPRESSION
Radius	`25e-3`
Base	Center
x-position	`61.5e-3`
y-position	`0`

12 In the same manner, create a second circle with the following properties:

OBJECT DIMENSIONS	EXPRESSION
Radius	`6e-3`
Base	Center
x-position	`61.5e-3`
y-position	`0`

13 Press the Shift key and click the **Rectangle/Square** button on the Draw toolbar.

14 In the dialog box that appears, enter the following rectangle properties; when done, click **OK**:

OBJECT DIMENSIONS	EXPRESSION
Width	`50e-3`
Height	`25e-3`
Base	Corner
x-position	`36.5e-3`
y-position	`-25e-3`

15 Create another rectangle with the following properties:

OBJECT DIMENSIONS	EXPRESSION
Width	`12e-3`
Height	`6e-3`
Base	Corner
x-position	`55.5e-3`
y-position	`-6e-3`

16 Select the circle C1 and the rectangle R1, then click the **Difference** button on the Draw toolbar.

17 Similarly, select the circle C2 and the rectangle R2, then click the **Difference** button.

18 Select the object CO1, then from the **Draw** menu select **Embed**.

19 In the dialog that appears, click **OK**.

20 Return to the **Geom2** geometry.

21 Select the object CO2, then from the **Draw** menu select **Extrude**.

22 In the dialog that appears, find the **Distance** edit field and type `-12.7e-3`.

23 Click **OK**.

24 Click the **Zoom Extents** button on the Main toolbar.

PHYSICS SETTINGS

Boundary Expressions

1 From the **Options** menu, select **Expressions>Boundary Expressions**.

2 Select Boundary 11, then enter the following expressions.

NAME	EXPRESSION
R	sqrt((x-0.0615)^2+y^2)
q_shoulder	(mu*F_n/A_s)*(R*omega)*flc1hs((T_melt-T)[1/K],5)

3 Select Boundaries 12 and 16, then enter the following expressions (on the third line of the table); when finished, click **OK**.

NAME	EXPRESSION
q_pin	mu/sqrt(3*(1+mu^2))*(r_pin*omega)*Ybar(T[1/K])[MPa]* flc1hs((T_melt-T)[1/K],5)

Note: The boundary expressions defined above include a smoothed step function, `flc1hs`, which models that the generation of friction heat is zero above the melting temperature of aluminum. Using this function is computationally more stable than multiplying the expressions by the logical expression `(T<T_melt)`.

Subdomain Settings

1 Choose **Physics>Subdomain Settings**.

2 Select Subdomains 1, 2 and 4, then click the **Load** button.

3 From the **Materials** list, select **Basic Material Properties>Aluminum**, then click **OK**.

4 Select Subdomain 3 and enter the following settings:

PROPERTY	VALUE
k (isotropic)	k_pin
ρ	rho_pin
C_p	Cp_pin

5 Click the **Convection** tab and select Subdomains 1, 2 and 4.

6 Select the **Enable convective heat transfer** check box.

7 In the **Matter state** list, select **Solid**.

8 In the leftmost **u** edit field for the *x* velocity, type `-u_weld`.

9 Select Subdomains 1 and 4, and click the **Infinite Elements** tab.

10 From the **Type of infinite element** list, select **Cartesian**. Then select the **Stretched in x direction** check box.

11 Enter the following properties in the $\mathbf{x_0}$, $\mathbf{y_0}$, and $\mathbf{z_0}$ edit fields:

PROPERTY	VALUE
x_0	61.5e-3
y_0	3e-3
z_0	6e-3

12 Select all subdomains and click the **Init** tab. Type `T0` in the **T(t$_0$)** edit field for the initial value.

13 Click **OK**.

Boundary Settings

1 From the **Physics** menu, select **Boundary Settings**.

2 Select the **Interior boundaries** check box.

3 Enter settings as in the following table; when finished, click **OK**.

SETTINGS	BOUNDARY 1	BOUNDARIES 3, 8, 20	BOUNDARY 4, 9, 21	BOUNDARY 11	BOUNDARIES 12, 16	BOUNDARY 23
Type	Insulation/ Symmetry	Heat flux	Heat flux	Heat flux	Heat source/ sink	Temperature
q_0				q_shoulder	q_pin	
h		h_downside	h_upside	0		
T_{inf}		T0	T0	273.15		
T_0						T0
Radiation type		Surface-to-ambient	Surface-to-ambient	None		
ε		epsilon	epsilon			
T_{amb}		T0	T0			

For the boundaries not mentioned in the table, the default setting (Insulation/ Symmetry) applies.

MESH GENERATION

From the **Mesh** menu, select **Initialize Mesh**.

COMPUTING THE SOLUTION

1 From the **Solve** menu, open the **Solver Parameters** dialog box.

2 Click the **Stationary** tab.

3 Type `60` in the **Maximum number of iterations** edit field.

4 Click **OK**.

5 From the **Solve** menu, choose **Solve Problem**.

POSTPROCESSING AND VISUALIZATION

The default plot shows a slice plot of the temperature field. To create Figure 3-19, which shows a slice plot and some temperature isosurfaces, follow these steps:

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2 Go to the **Slice** page and find the **Slice positioning** area.

3 In the **Number of levels** edit field for **x levels**, change the value to `0`.

4 From the **Colormap** list in the **Slice color** area, select **hot**.

5 In the **Vector with coordinates** edit field for **z levels**, type `1e-3`.

6 Click **Apply**.

7 On the **Isosurface** page, select the **Isosurface plot** check box.

8 In the **Vector with isolevels** edit field, type `300:20:980`.

9 From the **Colormap** list in the **Isosurface color** area, select **hot**.

10 Clear the **Color scale** check box.

11 Click **OK**.

Continuous Casting

Introduction

This example simulates the process of continuous casting of a metal rod from a melted state (Figure 3-21). To optimize the casting process in terms of casting rate and cooling, it is helpful to model the thermal and fluid dynamic aspects of the process. To get accurate results, you must model the melt flow field in combination with the heat transfer and phase change. The model includes the phase transition from melt to solid, both in terms of latent heat and the varying physical properties. The following model was originally developed by J. Fjellstedt (Outokumpu Copper, R&D).

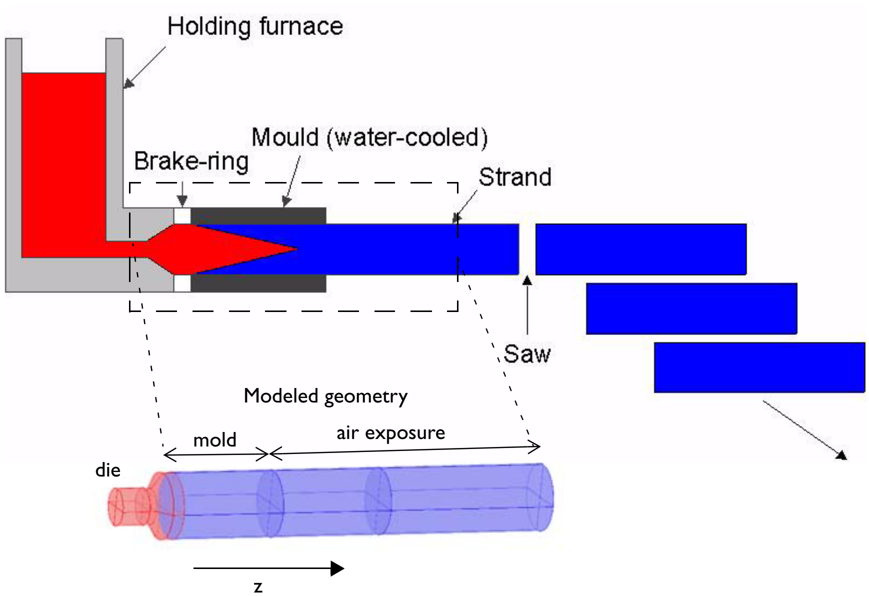


Figure 3-21: Continuous metal-casting process with an exploded view of the modeled section.

The model simplifies the rod's 3D geometry in Figure 3-21 to an axisymmetric 2D model in the *rz*-plane. Figure 3-22 shows the dimensions of the 2D geometry.

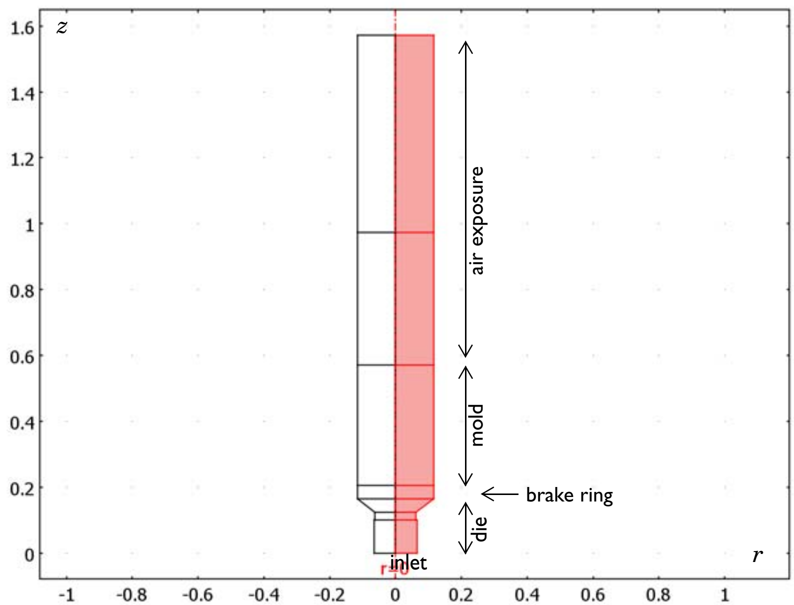


Figure 3-22: 2D axisymmetric model of the casting process.

As the melt cools down in the mold it solidifies. The phase transition releases latent heat, which the model includes. Furthermore, for metal alloys, the transition is often spread out over a temperature range. As the material solidifies, the material properties change considerably. Finally, the model also includes the "mushy" zone—a mixture of solid and melted material that occurs due to the rather broad transition temperature of the alloy and the solidification kinetics.

This example models the casting process as being stationary using two application modes: General Heat Transfer and Weakly Compressible Navier-Stokes.

Model Definition

The process operates at steady state, because it is a continuous process. The heat transport is described by the equation:

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

where k, C_p, and Q denote thermal conductivity, specific heat, and heating power per unit volume (heat source term), respectively.

As the melt cools down in the mold, it solidifies. During the phase transition, a significant amount of latent heat is released. The total amount of heat released per unit mass of alloy during the transition is given by the change in enthalpy, ΔH. In addition the specific heat capacity, C_p, also change considerably during the transition. As opposed to pure metals, an alloy generally undergoes a broad temperature transition zone, over several kelvin, in which a mixture of both solid and molten material co-exist in a "mushy" zone. To account for the latent heat related to the phase transition, replace C_p in the heat equation with $(C_p + \delta\Delta H)$, where ΔH is the latent heat of the transition, and δ is a Gaussian curve given by

$$\delta = \frac{\exp(-(T-T_{\mathrm{m}})^2/(\Delta T)^2)}{\Delta T\sqrt{\pi}}$$

Here T_{m} is the melting point and ΔT denotes the half-width of the curve, in this case set to 5 K, representing half the transition temperature span. The change in specific heat can be approximated by:

$$\Delta C_p = \frac{\Delta H}{T}$$

The change in specific heat also appears during the phase transition. You represent it using COMSOL Multiphysics' built-in smooth Heaviside step function `flc2hs` (for more details on `flc2hs`, see the *COMSOL Multiphysics Reference Guide*).

This example models the laminar flow using the Weakly Compressible Navier-Stokes application mode. The application mode describes the fluid velocity, $\mathbf{u}$, and the pressure, p, according to the equations:

$$\rho\frac{\partial\mathbf{u}}{\partial t} + \rho\mathbf{u}\cdot\nabla\mathbf{u} = \nabla\cdot[-p\mathbf{I} + \eta\,(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - (2\eta/3 - \kappa)(\nabla\cdot\mathbf{u})\mathbf{I}] + \mathbf{F}$$

$$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$$

where ρ is the density (in this case constant), η is the viscosity, and κ is the dilatational viscosity (here assumed to be zero). The source term, $\mathbf{F}$, is in this model used to dampen the velocity at the phase-change interface so that it becomes that of the solidified phase after the transition. The source term follows from the equation (see Ref. 1):

$$\mathbf{F} = \frac{(1-B)^2}{B^3+\varepsilon} A_{\mathrm{mush}}(\mathbf{u} - \mathbf{u}_{\mathrm{cast}})$$

where B is the volume fraction of the liquid phase; A_{mush} and ε represent arbitrary constants, (A_{mush} should be large and ε small to produce a proper damping); and $\mathbf{u}_{\mathrm{cast}}$ is the velocity of the cast rod. The fraction of liquid phase, B, is given by

$$B = \begin{cases} 1 & |(T > T_{\mathrm{m}} + \Delta T) \\ (T - T_{\mathrm{m}} + \Delta T)/(2\Delta T) & |(T_{\mathrm{m}} - \Delta T) \le T \le (T_{\mathrm{m}} + \Delta T) \\ 0 & |T < T_{\mathrm{m}} - \Delta T \end{cases}$$

Table 3-4 reviews the material properties in this model.

TABLE 3-4: MATERIAL PROPERTIES

PROPERTY	MELT	SOLID
ρ (kg/m^3)	8500	8500
C_p (J/(mol·K))	530	380
k (W/(m·K))	200	200
η (Ns/m^2)	0.0434	-

Furthermore, the melting temperature, T_{m}, and enthalpy, ΔH, are set to 1356 K and 205 kJ/(kg·K), respectively.

The model uses the parametric solver in combination with adaptive meshing to solve the problem efficiently.

Results and Discussion

The plots in Figure 3-23 display the temperature and phase distributions, showing that the melt cools down and solidifies in the mold region. Interestingly, the transition zone stretches out towards the center of the rod because of poorer cooling in that area.

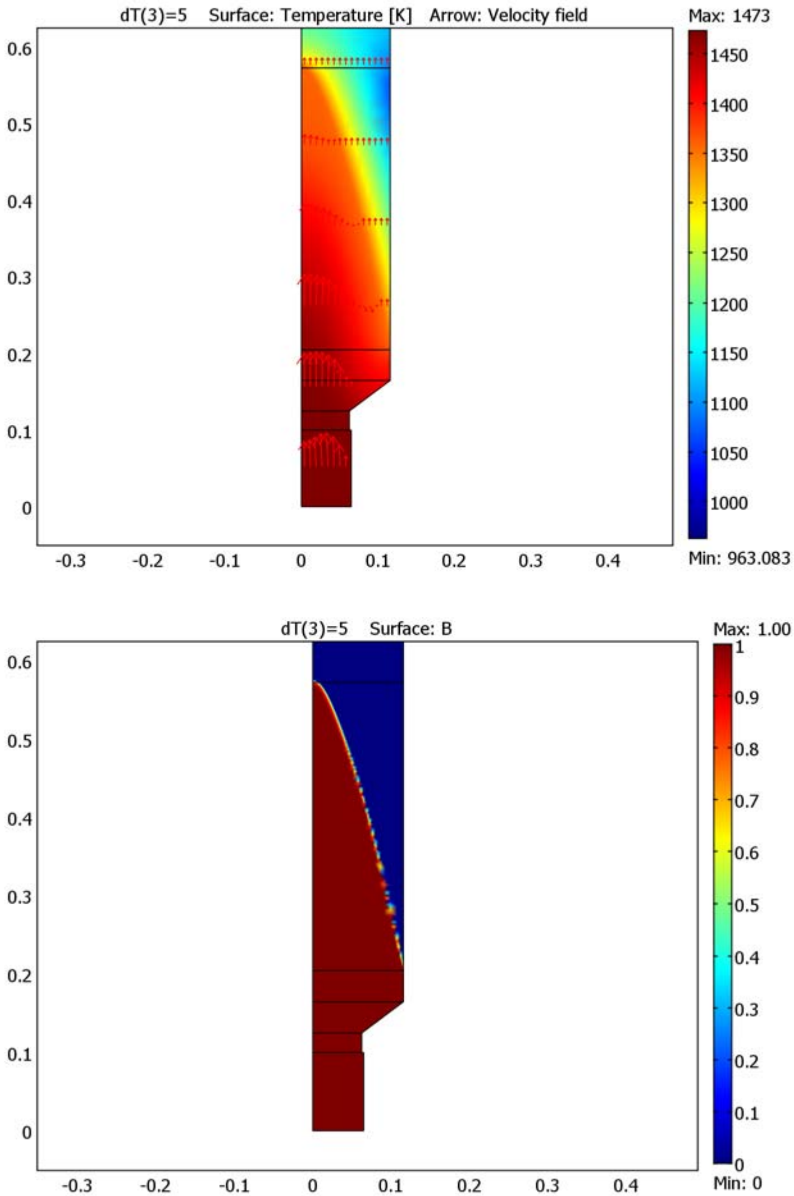


Figure 3-23: Temperature distribution (top) and fraction of liquid phase (bottom) in the lower part of the cast at a casting rate of 1.6 mm/s.

With the modeled casting rate, the rod is fully solidified before leaving the mold (the first section after the die). This means that the process engineers can increase the casting rate without running into problems, thus increasing the production rate.

The phase transition occurs in a very narrow zone although the model utilizes a transition half width, ΔT, of 5 K. In reality it would be even more distinct if a pure metal were being cast but somewhat broader if the cast material were an alloy with a wider ΔT.

It is interesting to study in detail the flow field in the melt as it exits the die. Figure 3-24 shows a zoom of this region.

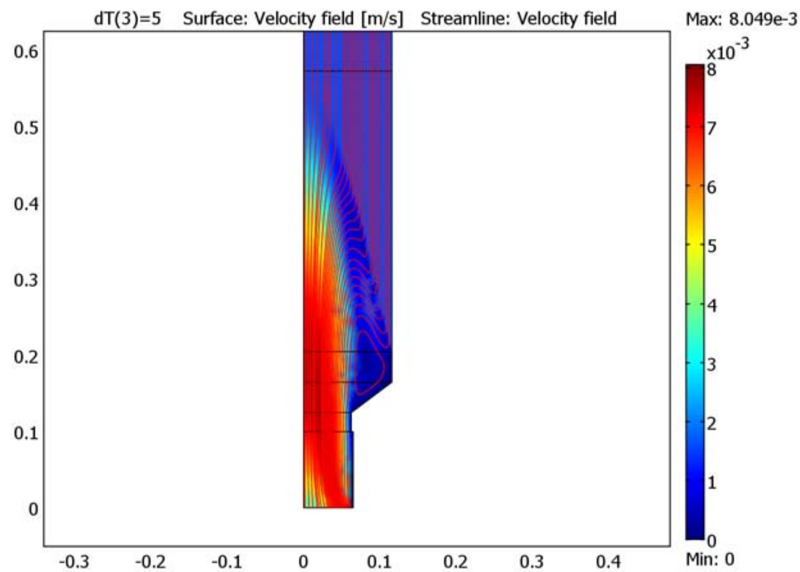


Figure 3-24: Velocity field with streamlines the lower part of the process.

Notice that there is disturbance in the streamlines close to the die wall resulting in a vortex. This eddy flow could create problems with nonuniform surface quality in a real process. Process engineers can thus use the model to avoid these problems and find an optimal die shape.

In order to help determine how to optimize process cooling, Figure 3-25 plots the conductive heat flux. It shows that the conductive heat flux is very large in the mould zone. This is a consequence of the heat released during the phase transition, which is cooled by the water-cooling jacket of the mould. An interesting phenomenon of the process is the peak of conductive heat flux appearing in the center of the flow at the transition zone.

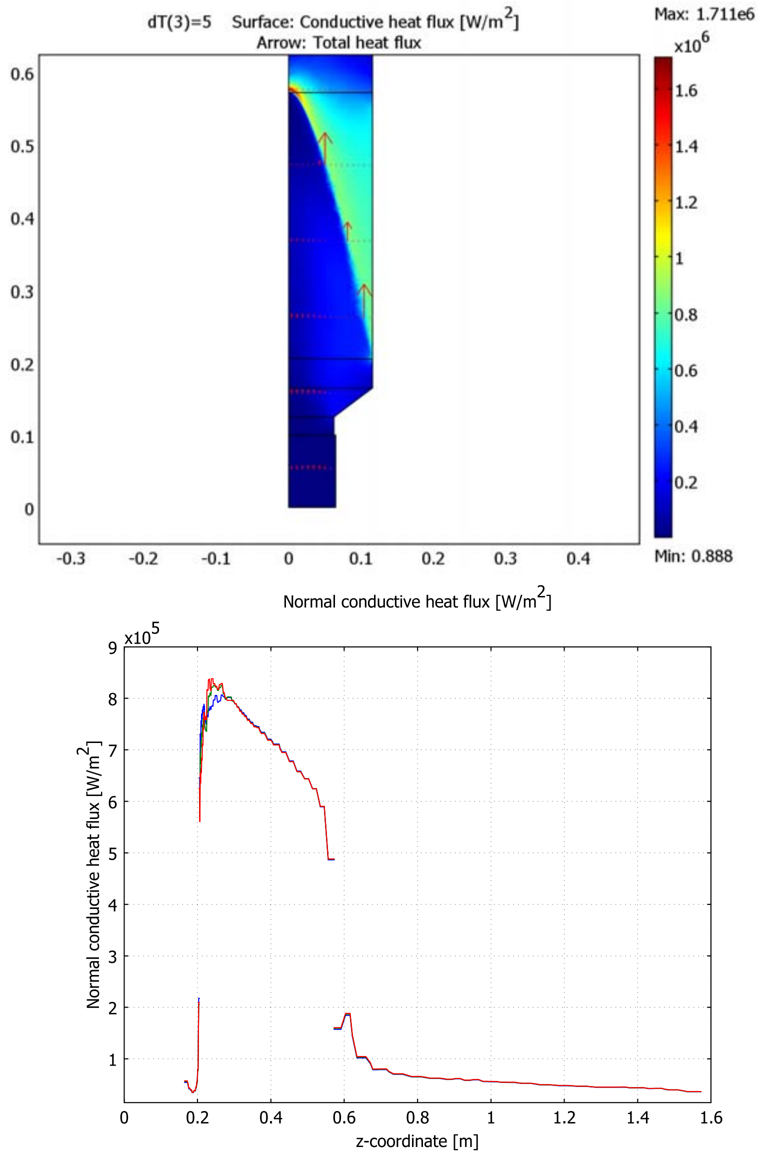


Figure 3-25: The cooling viewed as conductive heat flux in the domains (top), and along the outer boundary (the cooling zones) after the die (bottom).

Furthermore, by plotting the conductive heat flux at the outer boundary for the process as in the lower plot in Figure 3-25, you can see that a majority of the process cooling occurs in the mold. More interestingly, the heat flux varies along the mold wall length. This information can help in optimizing the cooling of the mold (that is, the cooling rate and choice of cooling method).

You solve the model using a built-in adaptive meshing technique. This is necessary because the transition zone—that is, the region where the phase change occurs—requires a fine discretization. Figure 3-26 depicts the final mesh of the model. Notice that the majority of the elements are concentrated to the transition zone.

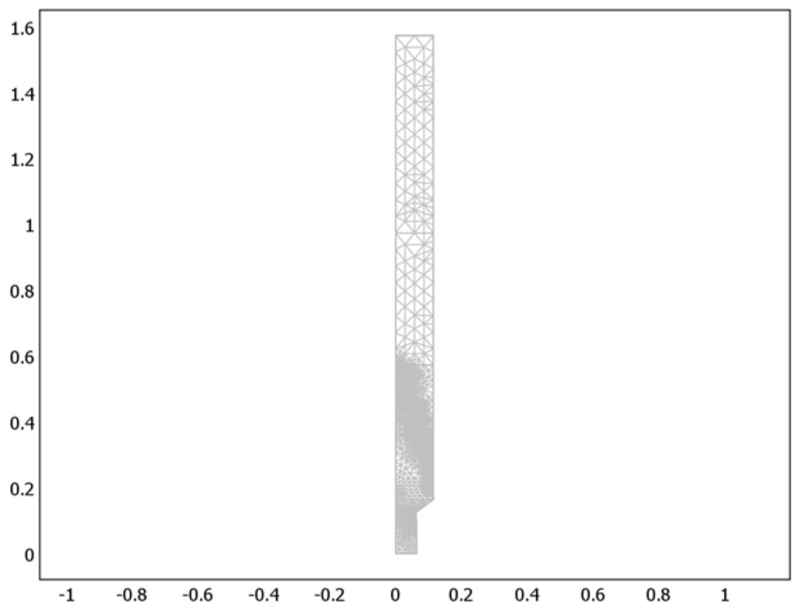


Figure 3-26: The final computational mesh, resulting from the built-in adaptive technique.

The adaptive meshing technique allows for fast and accurate calculations even if the transition width is brought down to a low value, such as for pure metals.

Reference

1. V.R. Voller, C. Prakash, *Int.J.Heat Mass Transfer*, vol. 30, pp. 1709–1719, 1987.

Model Library path: Heat_Transfer_Module/Process_and_Manufacturing/continuous_casting

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Open the **Model Navigator** and click the **New** tab. In the **Space dimension** list select **Axial symmetry (2D)**.
2. From the list of application modes, select **Heat Transfer Module>Fluid-Thermal Interaction>Non-Isothermal Flow>Steady-state analysis**.
3. Click **OK**.

OPTIONS AND SETTINGS

1. From the **Options** menu, select **Constants**. Define the following names and expressions; when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T0	300[K]	Ambient temperature
T_in	1473[K]	Melt inlet temperature
v_cast	1.6[mm/s]	Casting speed
T_m	1356[K]	Melting temperature
dT	20[K]	Temperature transition zone half width
dH	205[kJ/kg]	Latent heat

2. From the **Options** menu, select **Expressions>Scalar Expressions**. Define the following expressions; when finished, click **OK**.

NAME	EXPRESSION
D	exp(-(T-T_m)^2/(dT^2))/sqrt(pi*dT^2)
B	(T-T_m+dT)/(2*dT)*((T<=(T_m+dT))*(T>=(T_m-dT)))+(T>(T_m+dT))
Sr	(1-B)^2/(B^3+1e-3)*1e5[kg/(m^3*s)]*u
Sz	(1-B)^2/(B^3+1e-3)*1e5[kg/(m^3*s)]*(v-v_cast)
H	flc2hs(T-T_m,dT)
Cp1	380[J/(kg*K)]+dH/T_m*H

GEOMETRY MODELING

1 Create six rectangles. To do so, go to the **Draw** menu and select **Specify Objects>Rectangle**. Each time you open the dialog box, enter the following data for one of the rectangles, then click **OK**.

OBJECT	WIDTH	HEIGHT	BASE	r	z
R1	0.065	0.1	Corner	0	0
R2	0.0625	0.025	Corner	0	0.1
R3	0.11575	0.04	Corner	0	0.165
R4	0.11575	0.3675	Corner	0	0.205
R5	0.11575	0.4	Corner	0	0.5725
R6	0.11575	0.6	Corner	0	0.9725

2 Click the **Zoom Window** button on the Main toolbar, then click and drag in the drawing area to expand the viewing area between rectangles R2 and R3.

3 Draw two lines that join R2 and R3. To do so, click the **Line** button on the Draw toolbar, then click the right mouse button to complete each line.

4 Select all objects by pressing Ctrl+A.

5 From the **Draw** menu, select **Coerce To>Solid** to fill in the trapezoidal area between R2 and R3 and form one large solid.

PHYSICS SETTINGS

Subdomain Settings

1 From the **Multiphysics** menu, select the **Weakly Compressible Navier-Stokes** application mode.

1 From the **Physics** menu, select **Subdomain Settings**.

2 Select all subdomains, then enter the following expressions in the edit fields on the **Physics** page:

PARAMETER	EXPRESSION
η	0.0434
F_r	-Sr
F_z	-Sz

The default stabilization method for Navier-Stokes is GLS streamline diffusion and crosswind diffusion. For diffusive flows with strong source terms, using bubble elements, also known as mini-elements, is a good alternative.

3 Click the **Stabilization** tab.

4 Clear the **Streamline diffusion (GLS)** and the **Crosswind diffusion** check boxes, then click **OK**.

5 Click the **Element** tab, then from the **Predefined elements** list, select **p_1+p_1 (Mini)**.

6 Click the **Init** tab, then in the **z-velocity** edit field type `v_cast`. Click **OK**.

7 From the **Multiphysics** menu, select the **General Heat Transfer** application mode.

8 From the **Physics** menu, open the **Subdomain Settings** dialog box.

9 Select all subdomains.

10 On the **Init** page, go to the **Temperature** edit field and type `T_in`.

11 Click the **General** tab. Enter the following settings; when finished, click **OK**.

PARAMETER	EXPRESSION
k (isotropic)	200
ρ	8500
C_p	Cp1+D*dH

12 Click the **Convection** tab.

The predefined multiphysics coupling sets up the convective heat transfer part automatically (click the **Convection** tab if you want to verify the settings).

13 Click the **Element** tab, then from the **Predefined elements** list, select **Lagrange - Linear**.

14 Click the **Stabilization** tab.

15 Select the **Crosswind diffusion** checkbox, and from the **Elements of type** list select **Lagrange - Linear** to match the element type on the **Element** page.

16 Click **OK**.

Boundary Conditions

1 From the **Physics** menu open the **Boundary Settings** dialog box.

2 Set boundary conditions according to the following table. Note that h is different for Boundaries 20 and 21. When done, click **OK**.

SETTINGS	BOUNDARIES 1, 3, 5, 7, 9, 11, 13	BOUNDARY 2	BOUNDARY 15	BOUNDARIES 16–19	BOUNDARY 20	BOUNDARY 21	BOUNDARIES 22, 23
Type	Axial symmetry	Temperature	Convective flux	Insulation/ Symmetry	Heat flux	Heat flux	Heat flux
h					25	800	10
T_{inf}					T0	T0	T0

SETTINGS	BOUNDARIES 1, 3, 5, 7, 9, 11, 13	BOUNDARY 2	BOUNDARY 15	BOUNDARIES 16–19	BOUNDARY 20	BOUNDARY 21	BOUNDARIES 22, 23
T_0		T_in					
Radiation type		None			None	None	Surface-to-ambient
ε							0.8
T_{amb}							T0

3. In the **Multiphysics** menu, select **Weakly Compressible Navier-Stokes**.
4. From the **Physics** menu, open the **Boundary Settings** dialog box.
5. Enter the following settings; when finished, click **OK**.

SETTINGS	BOUNDARIES 1, 3, 5, 7, 9, 11, 13	BOUNDARY 2	BOUNDARIES 15, 21–23	BOUNDARIES 16–20
Boundary type	Symmetry boundary	Inlet	Outlet	Wall
Boundary condition	Axial symmetry	Pressure, no viscous stress	Velocity	No slip
u_0			0	
v_0			v_cast	
p_0		0		

MESH GENERATION

1. From the **Mesh** menu, select **Initialize Mesh**.
2. Click the **Refine Mesh** button on the Main toolbar.

COMPUTING THE SOLUTION

A three-step solution process calculates the solution. First you solve the problem using the parametric solver on the default mesh, gradually decreasing the value of `dT`. Then you use the adaptive solver to adapt the mesh. Finally, you use the Parametric solver to decrease `dT` further down to a value of 5.

1. From the **Solve** menu, open the **Solver Parameters** dialog box.
2. On the **General** page, select **Parametric** from the **Solver** list.
3. In the **Parameter name** edit field type `dT`, and in the **Parameter values** edit field type `300 100 50 20`.
4. On the **Stationary** page, set the **Maximum number of iterations** to `50`.
5. Click **Apply**.

6 Click the **Solve** button on the main toolbar to compute the first solution.

7 In the **Solver Parameters** dialog box, return to the **General** page and select the **Stationary** solver from the **Solver** list.

8 Select the **Adaptive mesh refinement** check box below the **Solver** list.

9 Click **Apply**.

10 Select **Solve>Restart** to start the adaptive solver based on the last solution.

11 To view the adapted mesh, choose **Mesh>Mesh Mode**.

You should now have a mesh resembling that in Figure 3-26, based on approximately 17,000 elements. You can view the statistics of the mesh by choosing **Mesh>Mesh Statistics**.

12 In the **Solver Parameters** dialog box, select the **Parametric** solver again.

13 Clear the **Adaptive mesh refinement** check box.

14 Change the **Parameter values** to `20 10 5`.

15 Click the **Parametric** tab and select the **Manual tuning of parameter step size**.

16 Set the **Initial step size** to `5`, the **Maximum step size** to `2.5`, and the **Maximum step size** to `5`.

17 Click the **Stationary** tab. Clear the **Highly nonlinear problem** check box.

18 Click **OK**.

19 To compute the last solution, click the **Restart** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To generate the upper plot in Figure 3-23, follow these steps:

1 From the **Postprocessing** menu, open the **Plot Parameters** dialog box.

2 Go to the **Surface** page, then in the **Predefined quantities** list select **Temperature**.

3 Click the **Arrow** tab. Select the **Arrow plot** check box at the top of the dialog box to enable this type of plot. On the **Subdomain Data** page, select **Velocity field** from the **Predefined quantities** list.

4 Click **Apply**.

5 Use the **Zoom Window** tool on the Main toolbar to zoom in on the geometry's lower half.

To generate the lower plot in Figure 3-23, continue with these steps:

1 Return to the **Plot Parameters** dialog box and clear the **Arrow plot** check box, then click the **Surface** tab.

2 On the **Surface Data** page, type B in the **Expression** edit field. (This variable represents the fraction of the volume in the liquid phase.)

3 Click **Apply**.

To generate Figure 3-24 execute the following instructions:

1 Still on the **Surface** page of the **Plot Parameters** dialog box, select **Velocity field** from the **Predefined quantities** list on the **Surface Data** page.

2 Click the **Streamline** tab. Select the **Streamline plot** check box.

3 On the **Streamline Data** page, select **Velocity field** from the **Predefined quantities** list. In the **Number of start points** edit field, type 33.

4 Click **Apply**.

You reproduce the upper plot in Figure 3-25 as follows:

1 In the **Plot Parameters** dialog box, clear the **Streamline plot** check box.

2 Click the **Arrow** tab. Select **Total heat flux** from the **Predefined quantities** list.

3 Click the **Surface** tab. Select **Conductive heat flux** from the **Predefined quantities** list on the **Surface Data** page.

4 Click **OK** to generate the plot and close the **Plot Parameters** dialog box.

Finally, generate the lower plot in Figure 3-25 with the following steps:

1 From the **Postprocessing** menu, select **Domain Plot Parameters**. Go to the **Line/ Extrusion** page.

2 From the **Predefined quantities** list, select **Normal conductive heat flux**.

3 In the **x-axis data** area, click first the lower option button and then the **Expression** button.

4 In the **X-Axis Data** dialog box, type **z** in the **Expression** edit field and then click **OK**.

5 Select Boundaries 20–23, then click **OK**.

Turbulent Flow Through a Shell-and-Tube Heat Exchanger

Introduction

This model describes a part of a shell-and-tube heat exchanger (see Figure 3-27), where hot water enters from above. The cooling medium, which is also commonly water, flows through the pipes and enters from the side. The tubes are assumed to be made of stainless steel.

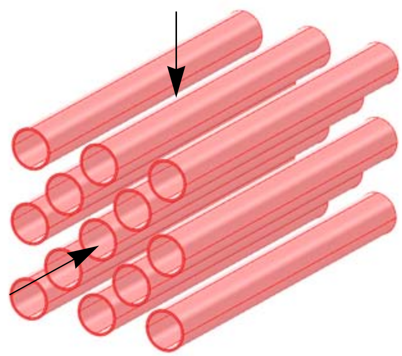


Figure 3-27: A tube bundle from a shell-and-tube heat exchanger. The arrows indicate the flow directions.

Assuming that the cooling water is in abundant supply, the flow through the pipes has a constant temperature. Under that assumption, you can model this heat exchanger by a 2D model as shown in Figure 3-28, and the corresponding 2D domain appears in

Figure 3-29. Note that the pipe interiors are not part of the domain because the model assumes the temperature to be constant there.

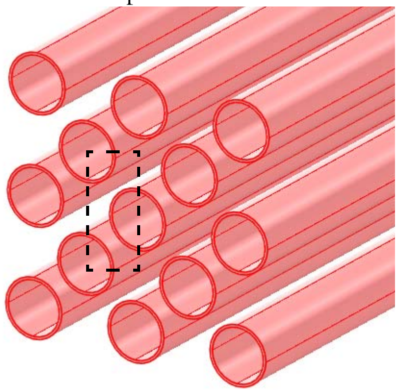


Figure 3-28: The dashed line marks the model region, which is shown in Figure 3-29.

THE NEED FOR A TURBULENCE MODEL

The characteristic of a flow is often described by the Reynolds number, which is defined as

$$\mathrm{Re} = \frac{\rho U L}{\eta}$$

where U is a velocity scale and L is a length scale. If the Reynolds number is low, no turbulence model is needed. If, on the other hand, the Reynolds number is high, then the flow is dominated by convection, and a turbulence model is necessary.

In this case, a suitable velocity scale is the mean inlet velocity, which is 0.5 m/s, and L is set to the pipe diameter. Then, using standard values for water for the density and viscosity, the equation gives an approximate Reynolds number of 50,000, which is high enough to warrant the use of a turbulence model. See Ref. 4 for more information on flow regimes for different Reynolds numbers.

The following example demonstrates how to model a conjugate heat transfer problem with COMSOL Multiphysics, using the Turbulent Fluid-Thermal Interaction predefined multiphysics coupling from the Heat Transfer Module. It also demonstrates how to generate a fully developed flow field when you know the mass flow rate but not the pressure drop.

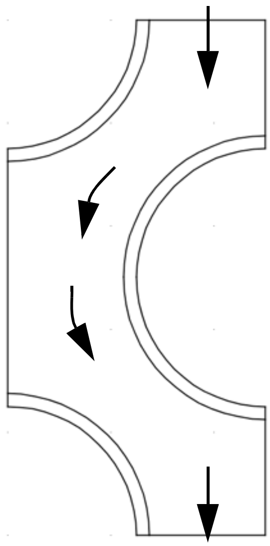


Figure 3-29: The modeled 2D region. The arrows indicate the flow.

Model Definition

SOLID AND FLUID HEAT TRANSFER—INCLUDING THE FLUID DYNAMICS

The governing equations in this model are:

- Reynolds Averaged Navier-Stokes (RANS) equations and a Wilcox revised k-ω turbulence model from 1998 in the water domain.
- Heat transport equations in the water and the solid (steel) tube walls.

The Turbulent Fluid–Thermal Interaction predefined multiphysics coupling sets up the appropriate application modes together with applicable couplings, making it easy to model the fluid-thermal interaction.

Temperature dependent properties for water and steel can be loaded from the built-in material library. It is necessary to correct the fluid's thermal conductivity to take into account the effect of mixing due to eddies. The turbulence results in an effective thermal conductivity, k_{eff}, according to the equation

$$k_{\text{eff}} = k + k_T \qquad k_T = \frac{C_p \eta_T}{Pr_T}$$

where k is the fluid's physical thermal conductivity and k_T is the turbulent conductivity. η_T denotes the turbulent dynamic viscosity, and C_p is the heat capacity. Pr_T is the turbulent Prandtl number. It is easy to obtain the effective conductivity in COMSOL Multiphysics by using the 'default' group in the fluid domain. In this group, the variable for turbulent conductivity is already given in the heat transfer application mode for the fluid. Be careful not to confuse k in the meaning of thermal conductivity with k in the meaning of turbulent kinetic energy.

Figure 3-30 depicts the model with its boundary conditions.

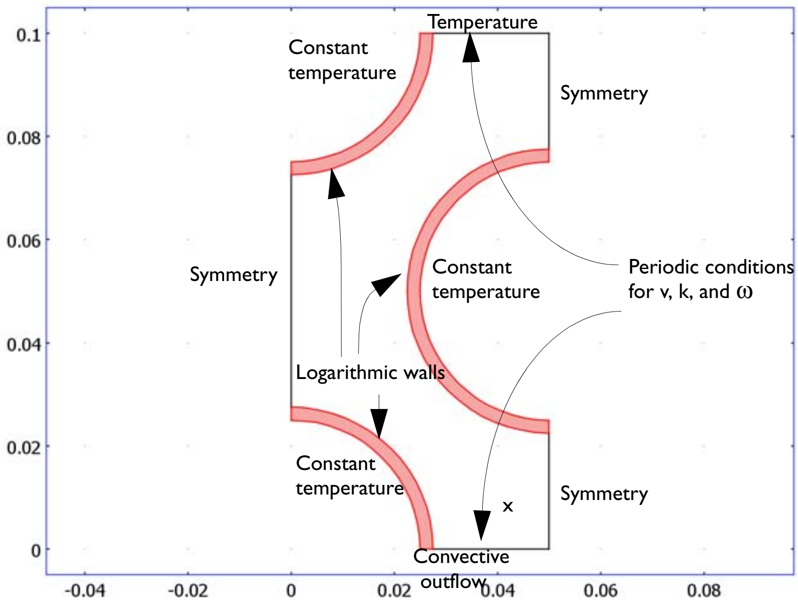


Figure 3-30: Modeled 2D geometry with boundary conditions.

The boundary conditions describing the problem are:

- k-ω equations in the fluid domain
  - Specified mass flow through the domain.
  - Pressure difference between inlet and outlet given by the mass flow
  - Normal flow at the inlet and outlet
  - Stream-wise periodic conditions for v, k, and ω.
  - Symmetry at the region borders
  - Logarithmic wall function at the pipes' surface boundaries
- Heat transport equations
  - 50 °C temperature at the inlet
  - Convection-dominated transport at the outlet
  - Symmetry (thermal insulation) at the region borders
  - Thermal wall function at the pipe/water interfaces
  - Fixed temperature at the inside of the heat pipes

The periodicity of the flow is important because you are modeling a part of the heat exchanger where the flow is fully developed. It is hard to make a periodic configuration converge from a homogeneous initial guess, however, and therefore, an initial calculation with constant inlet velocity and fixed outlet pressure is first performed.

The logarithmic wall function boundary condition for turbulent flow is used to model the solid-fluid interfaces. An algebraic relationship—the logarithmic wall function—describes the momentum transfer at the solid-fluid interface. This means that the solid-fluid boundaries in the model actually represent lines within the logarithmic regions of the boundary layers. Similar to the fluid velocity, the temperature is not modeled in innermost part of the boundary layer. Instead of assuming continuity of the temperature across the layer, a thermal "wall function" is used. There is a jump in temperature from the solid surface to the fluid due to the omitted innermost part of the boundary layer. The predefined group for the wall domains defines this wall function in the following way.

To achieve the thermal wall function, the model uses two heat transfer application modes: one for the solid, and one for the fluid. These are connected through a heat flux boundary condition, the thermal wall function. This means that the resistance to heat transfer through the innermost part of the boundary layer is related to that for momentum transfer for the fluid. The heat flux, q, is determined by the equation

$$q = \frac{\rho C_p C_\mu^{1/4} k_w^{\ 1/2}(T_w - T)}{T^+}$$

where ρ and C_p are the fluid's density and heat capacity, respectively; C_μ is a constant of the turbulence model; and k_w is the value of the turbulent kinetic energy at the wall. T_w equals the temperature of the solid at the wall, while T is the temperature of the fluid on the other side of the omitted laminar sublayer. The dimensionless quantity $T^+ = T^+(\delta_w^{\ +})$ is the dimensionless temperature and depends on the dimensionless wall offset, $\delta_w^{\ +}$.

Results for the Flow/Heat Model

Figure 3-31 depicts the temperature distribution and velocity streamlines. As the plot shows, the flow field is periodic in the y direction. This is important because the heat transfer is strongly influenced by the details of the velocity field. Observe the low-temperature zones behind the pipes created by the recirculation zones there.

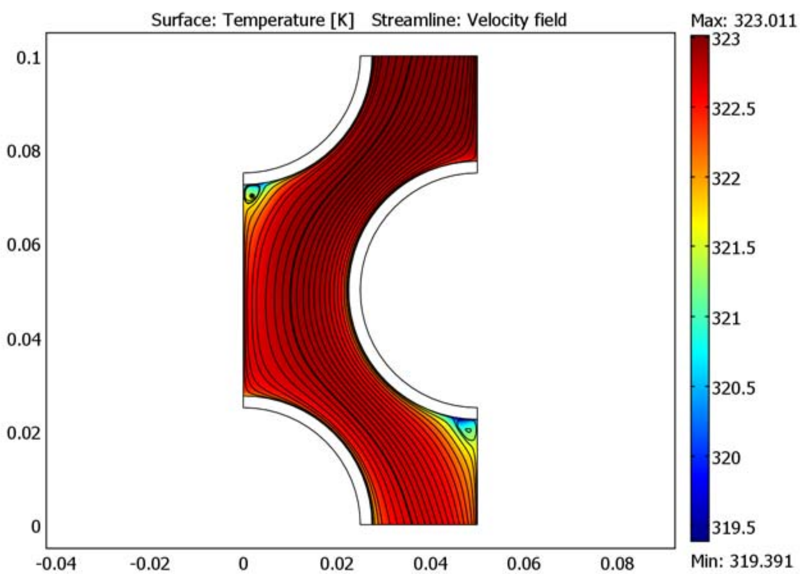


Figure 3-31: Temperature distribution for the periodic boundary case with a specified mean mass flow.

References

1. B.E. Launder and D.B. Spalding, "The Numerical Computation of Turbulent Flows," *Computer Methods in Applied Mechanics*, vol. 3, pp. 269–289, 1974.

2. D.C. Wilcox, *Turbulence Modeling for CFD, 2nd ed*, DCW Industries, 2000.

3. H. K. Versteeg and W. Malalasekera, *An Introduction to Computational Fluid Dynamics*, Prentice Hall, 1995.

4. J. R. Welty, C. E. Wicks, and R. E. Wilson, *Fundamentals of Momentum, Heat and Mass Transfer, 3rd ed*, John Wiley & Sons, Inc., 1984.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/turbulent_heat_exchanger`

Modeling Using the Graphical User Interface

The COMSOL Multiphysics implementation is straightforward using the Heat Transfer Module. You build the model in several steps to ensure accurate results.

MODEL NAVIGATOR

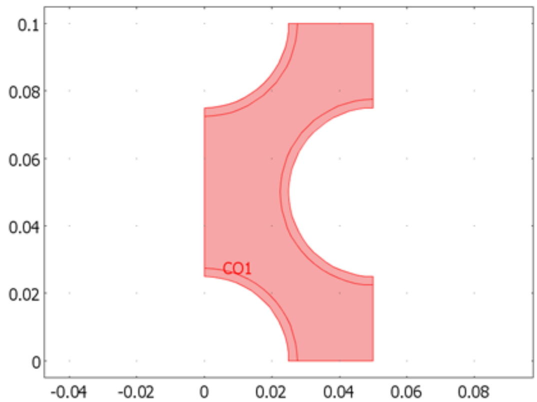
1. Open the **Model Navigator**, and in the **Space dimension** list select **2D**.
2. In the list of application modes select **Heat Transfer Module>Fluid-Thermal Interaction>Turbulent Non-Isothermal Flow, k-ω**.
3. Click **OK**.

GEOMETRY MODELING

To save time, load the CAD model as a COMSOL Multiphysics Geometry file:

1. From the **File** menu select **Import>CAD Data From File**. A dialog box opens.
2. Select **All 2D CAD files** from the **Files of type** list.
3. Browse to the folder `Models/Heat_Transfer_Module/Process_and_Manufacturing` in the COMSOL Multiphysics installation directory. Find and select the file `turbulent_heat_exchanger.mphbin`.
4. Click **Import** to load the model.

You should now see the following geometry:



CONSTANTS, EXPRESSIONS, AND VARIABLES

1 From the **Options** menu open the **Constants** dialog box.

2 Specify the following names, expressions, and descriptions (optional); when done, click **OK**.

NAME	EXPRESSION	DESCRIPTION
T_in	323[K]	Inflow temperature
T_pipe	278[K]	Pipe temperature
v_in	-0.5[m/s]	Inflow velocity
rho0	988[kg/m^3]	Reference density, water
L_in	0.025[m]	Width of inflow boundary
mf_in	L_in*rho0*v_in	Mass inflow

PHYSICS AND BOUNDARY SETTINGS

You first make a computation without periodic boundary conditions. This yields a velocity field that can be used as an initial guess for the periodic case.

1 Make sure that the k-ω Turbulence Model (chns) application mode is the active application mode.

2 Choose **Physics>Subdomain Settings**. Select Subdomains 1, 3, and 4 (the pipes).

3 Select **Solid domain** from the **Group** list.

4 Select Subdomain 2 (the fluid). Click the **Load** button to open the **Materials/ Coefficients Library** dialog box.

5 From the **Materials** tree, choose **Liquids and Gases>Liquids>Water**. Click **OK**.

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

7 Choose **Physics>Boundary Settings**.

8 Apply the following boundary settings; when done, click **OK**.

SETTINGS	BOUNDARY 7	BOUNDARIES 2, 8, 11	BOUNDARIES 13, 14, 16, 17	BOUNDARY 6
Object	Inlet	Symmetry boundaries	Pipe surfaces	Outlet
Boundary type	Inlet	Symmetry boundary	Wall	Outlet
Boundary condition	Velocity		Logarithmic wall function	Pressure
u_0	0			
v_0	`v_in`			
p_0				0

Now set up the parameters for the heat transfer.

1 In the **Multiphysics** menu, select the first **General Heat Transfer** application mode (**htgh**).

2 Choose **Physics>Subdomain Settings**.

3 Select Subdomains 1, 3, and 4. Select **Solid domain** from the **Group** list.

4 Select Subdomain 2. From the **Library material** list select **Water**.

5 Click **OK**.

6 From the **Physics** menu, open the **Boundary Settings** dialog box.

7 Specify boundary settings according to the following table. When done, click **OK**.

SETTINGS	BOUNDARY 7	BOUNDARIES 2, 8, 11	BOUNDARIES 13, 14, 16, 17	BOUNDARY 6
Object	Inlet	Symmetry boundaries	Pipe surfaces	Outlet
Group			wall	
Boundary condition	Temperature	Insulation/ Symmetry		Convective flux
T_0	`T_in`			

8 In the **Multiphysics** menu, select the second **General Heat Transfer** application mode (**htgh2**).

9 Choose **Physics>Subdomain Settings**.

10 For Subdomains 1, 3, and 4, select **Solid domain** from the **Group** list.

11 Click the **Load** button to open the **Materials/Coefficients Library** dialog box.

12 From the **Materials** tree, select **Basic Material Properties>Steel AISI 4340**. Click **OK**.

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

14 Open the **Physics>Boundary Settings** dialog box and specify the following boundary conditions. When done, click **OK**.

SETTINGS	BOUNDARIES 12, 15, 18, 19	BOUNDARIES 13, 14, 16, 17
Object	Pipe inner temperature	Pipe surfaces
Group		wall
Boundary condition	Temperature	
T_0	`T_pipe`	

MESH GENERATION

1 Click the **Initialize Mesh** button on the Main toolbar.

2 Click the **Refine Mesh** button on the Main toolbar once to generate the final mesh.

COMPUTING THE SOLUTION

The solution procedure involves a first solution step that solves a given inlet velocity. This yields a good initial condition for the final calculation with periodic boundary conditions.

1 Click the **Solve** button on the Main toolbar.

2 Choose **Solve>Solver Manager**.

3 On the **Initial Value** page, click the **Store Solution** button.

4 Click the **Current solution** option button in the **Initial value** area.

5 Click **OK**.

POSTPROCESSING AND VISUALIZATION

1 To generate Figure 3-32 open the **Postprocessing>Plot Parameters** dialog box.

2 On the **General** page, clear the **Surface** check box, then select the **Contour** check box.

3 On the **Contour** page, select **Velocity field** in the **Predefined quantities** list and click **OK**.

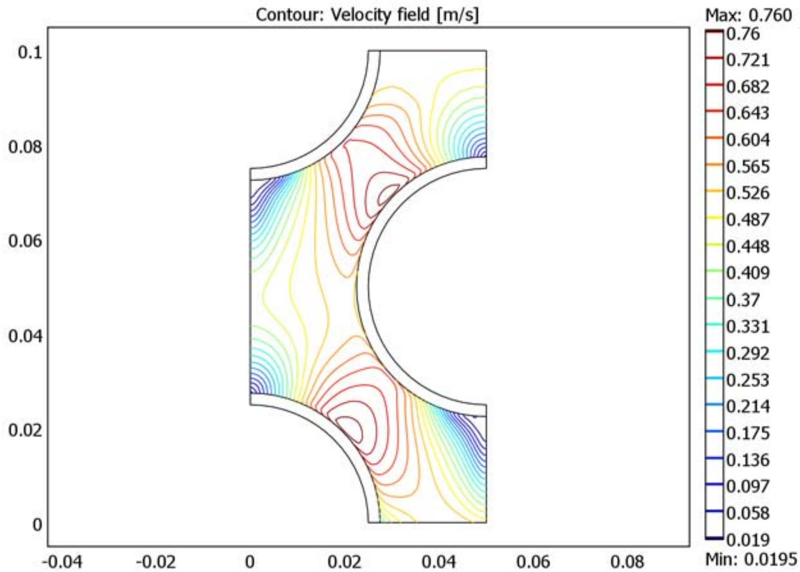


Figure 3-32: Velocity contours for a uniform inflow velocity.

In Figure 3-32 it is clear that the velocity profile on the outflow boundary is not uniform; instead, it varies considerably. Because the heat exchanger system is periodic in its structure, the velocity field should be periodic. The next step is therefore to add periodic boundary condition for the y-velocity, v, and for k and ω. The mass flux is controlled by an integral constraint implemented in the **Global Equations** dialog box.

1 Open the **Physics>Periodic Conditions>Periodic Boundary Conditions** dialog box, then select Boundary 7. Select the first row in the **Expression** column and type `v`. Type `logk` on the second row in the same column and `logw` on the third row.

2 On the **Destination** page select Boundary 6.

3 Type `logw` in the **Expression** edit field and select the **Use selected boundaries as destination** check box.

   The following two steps, Steps 4 and 5, specify the relative orientation of the fields on the source and destination boundaries.

4 On the **Source Vertices** page add Vertices 10 and 16 to the **Source vertices** list in this order by selecting them in the **Vertex selection** list and then clicking the **>>** button.

5 On the **Destination Vertices** page add Vertices 9 and 11 to the **Destination vertices** list. This identifies Vertex 10 with Vertex 9, and Vertex 16 with Vertex 11.
6 On the **Destination** page select **pconstr2** from the **Constraint name** list.
7 Type `logk` in the **Expression** edit field and select the **Use selected boundaries as destination** check box.
8 Repeat Steps 4 and 5.
9 On the **Destination** page select **pconstr1** from the **Constraint name** list.
10 Type `v` in the **Expression** edit field and select the **Use selected boundaries as destination** check box.
11 Repeat Steps 4 and 5.
12 Click **OK**.

The next step is to add an integration coupling variable that evaluates the total mass flux through the inlet. It is later used in an ODE to calculate an inlet pressure.

1 In the **Options** menu select **Integration Coupling Variables>Boundary Variables**, and choose Boundary 7.
2 Select the first row in the list. Type `mf` in the **Name** edit field and `rho_chns*v` in the **Expression** edit field. Click **OK**.

The next step is to add a global equation that constrains the mass flux. The **Global Equations** variable controls the inflow pressure level and yields the desired mass flow, `mf_in`. Also, a point pressure constraint must be added to keep the absolute pressure level fixed.

1 Open the **Physics>Global Equations** dialog box.
2 On the **States** page, select the first row. Type `p_in` in the **Name (u)** column, then enter `mf-mf_in` in the **Equation f(u,ut,utt,t)** column.
3 Click **OK** to close the dialog box.
4 In the **Multiphysics** menu, select the **k-ω Turbulence Model (chns)** application mode.
5 Choose **Physics>Boundary Settings**.
6 Select the inlet boundary (Boundary 7).
7 Change the **Boundary type** from **Inlet** to **Stress**. Set the **Boundary condition** to **Normal stress** and set $\mathbf{f_0}$ to `p_in`.
8 Click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 Add `p_in` as a **Component** of **Group 1**.

3 Click **OK**.

4 Click the **Solve** button on the Main toolbar.

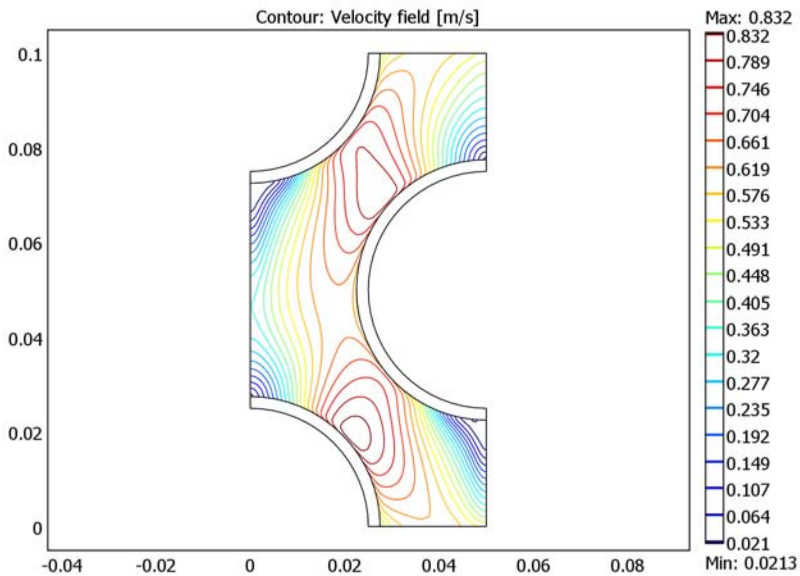


Figure 3-33: Velocity contours for periodic velocity conditions.

From Figure 3-33 it is now evident that the inflow and outflow velocity profiles match. In addition, the velocity is higher close to the pipe at the inflow region. This is clearly important when it comes to heat transfer, and it is a considerable improvement when compared to the uniform inflow profile.

POSTPROCESSING

1 To generate Figure 3-31, open the **Postprocessing>Plot Parameters** dialog box.

2 On the **General** page, clear the **Contour** check box and select the **Surface** and **Streamline** check boxes.

3 On the **Surface** page, choose **General Heat Transfer (htgh)>Temperature** in the **Predefined quantities** list.

4 On the **Streamline** page, choose **k-ω Turbulence model (chns)>Velocity field** in the **Predefined quantities** list. Set the **Streamline plot type** to **Magnitude controlled**.

5 Click the **Line Color** tab, then click the **Uniform color** option button. Click the **Color** button and select black, then click **OK**.

6 Click the **Advanced** button. Select the **Normalize vector field** check box, then click **OK** to close the **Advanced Streamline Parameters** dialog box.

7 Click **OK** to close the **Plot Parameters** dialog box and generate the plot.

Fluid-Structure Interaction in Aluminum Extrusion

Introduction

In massive forming processes like rolling or extrusion, metal alloys are deformed in a hot solid state with material flowing under ideally plastic conditions. Such processes can be simulated effectively using computational fluid dynamics, where the material is considered as a fluid with a very high viscosity that depends on velocity and temperature. Internal friction of the moving material acts as a heat source, so that the heat transfer equations are fully coupled with those ruling the fluid dynamics part. This approach is especially advantageous when large deformations are involved.

This model is adapted from a benchmark study in Ref. 1. The original benchmark solves a thermal-structural coupling, because it is common practice in the simulation of such processes to use specific finite element codes that have the capability to couple the structural equations with heat transfer. The alternative scheme discussed here couples non-Newtonian flow with heat transfer equations. In addition, because it is useful to know the stress in the die due to fluid pressure and thermal loads, the model adds a structural mechanics analysis to the other two.

The die design is courtesy of Compes S.p.A., while the die geometry, boundary conditions, and experimental data are those of Ref. 1.

Note: This model requires the Chemical Engineering Module, the Heat Transfer Module, and the Structural Mechanics Module. In addition, the model uses the Material Library.

Model Definition

The model considers steady-state conditions, assuming a billet of infinite length flowing through the die. In the actual process, the billet is pushed by the ram through the die and its volume is continuously reducing.

Figure 3-34 shows the original complete geometry with four different profiles. To have a model with reasonable dimensions, consider only a quarter of the original geometry. The simplification involved in neglecting the differences between the four profiles does not affect the numerical scheme proposed. Figure 3-35 shows the resulting model geometry.

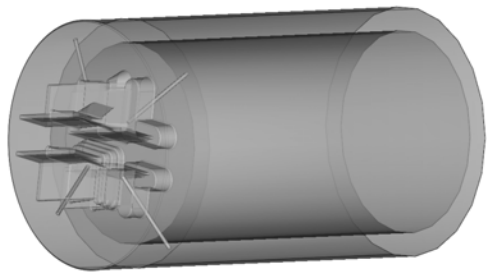


Figure 3-34: Original benchmark geometry.

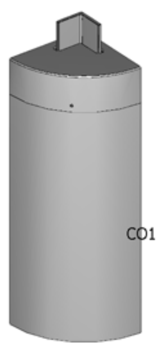


Figure 3-35: Quarter of the original geometry considered in the model.

MATERIAL PROPERTIES

The documentation associated with the benchmark (Ref. 1) serves as the data source for properties of the two main materials: AISI steel for the die and the container (the ram here is not considered) and aluminum for the billet.

Structural Analysis

Because only the steel part is active in the structural analysis, consider a simple linear elastic behavior where the elastic properties are those of the material H11 mod (AISI 610) that can be found in the COMSOL Multiphysics Material Library.

Heat Transfer Analysis

The documentation associated with the benchmark suggested to use for aluminum and for steel the following properties:

ALUMINUM	VALUE	DESCRIPTION
K_{al}	210 N/(s·K)	Conductivity
ρ_{al}	2700 kg/m^3	Density
C_{pal}	2.94 N/(mm^2·K)/ρ_{al}	Specific heat

STEEL	VALUE	DESCRIPTION
K_{fe}	24.33 N/(s·K)	Conductivity
ρ_{fe}	7850 kg/m^3	Density
C_{pfe}	4.63 N/(mm^2·K)/ρ_{fe}	Specific heat

Non-Newtonian Flow

The properties of the aluminum were experimentally determined and then checked using literature data for the same alloy and surface state. However the benchmark proposes an experimental constitutive law, suited for the structural mechanics codes usually used to simulate such processes, in the form of the flow stress data. For this model this requires a recalculation of the constitutive law to derive a general expression for the viscosity. The equivalent von Mises stress, σ_{eqv}, can be defined in terms of the total contraction of the deviatoric stress tensor as

$$\sigma_{eqv} = \sqrt{\frac{3}{2}\tau:\tau}$$

or, using $\tau = 2\eta\dot{\varepsilon}$ where $\dot{\varepsilon}$ is the strain rate and η is the viscosity, as

$$\sigma_{eqv} = \sqrt{6\eta^2\varepsilon:\varepsilon} \qquad (3\text{-}1)$$

Introducing the equivalent strain rate

$$\dot{\phi}_{eqv} \equiv \sqrt{\frac{2}{3}\varepsilon:\varepsilon}$$

Equation 3-1 can be expressed as

$$\sigma_{eqv} = 3\eta\dot{\phi}_{eqv}$$

The strain rate tensor is defined as (Ref. 2)

$$\dot{\varepsilon} = \frac{\nabla \mathbf{u} + (\nabla \mathbf{u})^T}{2} = \frac{1}{2}\dot{\gamma}$$

Equation 5-26 on page 142 in the *Chemical Engineering Module User's Guide* states that the shear rate $\dot{\gamma}$ is defined as

$$\dot{\gamma} = |\dot{\gamma}| = \sqrt{\frac{1}{2}\dot{\gamma}:\dot{\gamma}}$$

so that

$$\phi_{\text{eqv}} = \frac{1}{\sqrt{3}}\dot{\gamma}$$

The flow rule

$$\sigma_{\text{eqv}} = \kappa_{\text{f}}$$

states that plastic yielding occurs if the equivalent stress, σ_{eqv}, reaches the flow stress, κ_{f}. The viscosity is defined as (see Ref. 2 for further details)

$$\eta = \frac{\kappa_{\text{f}}}{3\phi_{\text{eqv}}}$$

The organizers of the benchmark propose specific flow-stress data expressed in terms of a generalized Zener-Hollomon function

$$\eta = \frac{\operatorname{asinh}\left(\left(\frac{Z}{A}\right)^{\frac{1}{n}}\right)}{\sqrt{3}\alpha\dot{\gamma}}$$

where $A = 2.39 \cdot 10^8\ \text{s}^{-1}$, $n = 2.976$, $\alpha = 0.052\ \text{MPa}^{-1}$, and

$$Z = \frac{1}{\sqrt{3}}\dot{\gamma}e^{\left(\frac{Q}{RT}\right)}$$

with $Q = 153$ kJ/mol and $R = 8.314$ J/(K·mol).

SOURCES, INITIAL CONDITIONS, AND BOUNDARY CONDITIONS

Structural Analysis

Because the model geometry is a quarter of the actual geometry, use symmetric boundary conditions for the two orthogonal planes. On the external surfaces of the die, apply roller boundary conditions because in reality other dies, not considered here, are present to increase the system's stiffness.

The main loads are the thermal loads from the heat transfer analysis and pressures from the fluid dynamics analysis.

Heat Transfer Analysis

For the billet, use a volumetric heat source related to the viscous heating effect.

The external temperature of the ram and the die is held constant at 450 °C. The ambient temperature is 25 °C. For the heat exchange between aluminum and steel, use the heat transfer coefficient of 11 N/(s·mm·K). Also consider convective heat exchange with air outside the profiles with a fixed convective heat transfer coefficient of 15 $W/(m^2 \cdot K)$.

Apply initial temperatures as given in the following table:

PART	VALUE
Ram	380 °C
Container	450 °C
Billet	460 °C
Die	404 °C

Non-Newtonian Flow

At the inlet, the ram moves with a constant velocity of 0.5 mm/s. Impose this boundary condition by simply applying a constant inlet velocity. At the outlet, a normal stress condition with zero external pressure applies. On the surfaces placed on the two symmetry planes, use symmetric conditions. Finally, apply slip boundary conditions on the boundaries placed outside the profile.

Results and Discussion

The general response of the proposed numerical scheme, especially in the zone of the profile, is in good accordance with the experience of the designers. A comparison between the available experimental data and the numerical results of the simulation shows good agreement.

On the basis of the results from the simulation, the engineer can improve the preliminary die design by adjusting relevant physical parameters and operating conditions. For this purpose, the slice plot in Figure 3-36 showing the temperature field inside the profile gives important information. Furthermore, the combined streamline and slice plot in Figure 3-37 reveals any imbalances in the velocity field that could result in a crooked profile. A proper design should also ensure that different parts of the profile travel at the same speed. In Figure 3-38 you can see von Mises equivalent strain in the steel part considering the thermal load and also the pressure due to the presence of the fluid.

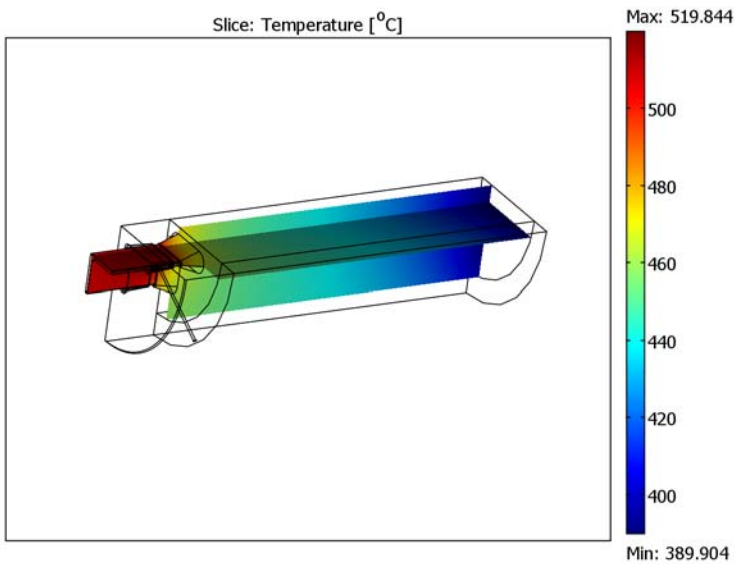


Figure 3-36: Temperature distribution in the billet.

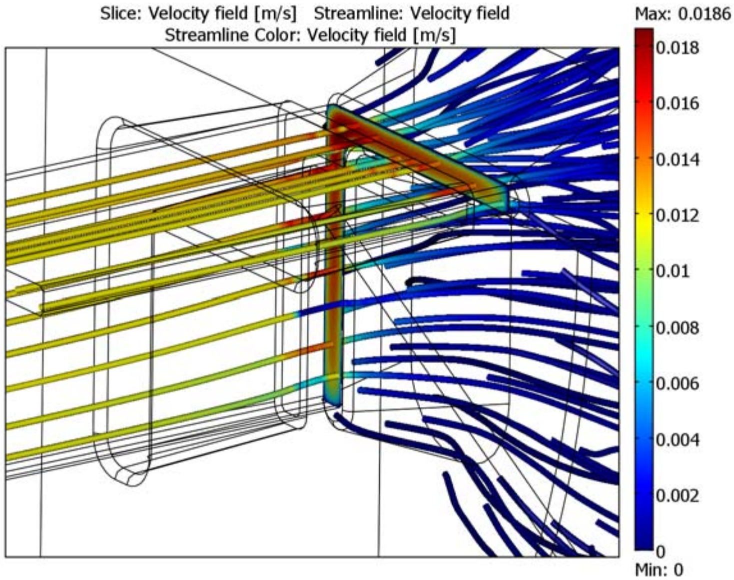


Figure 3-37: Velocity field and streamlines at the profile section.

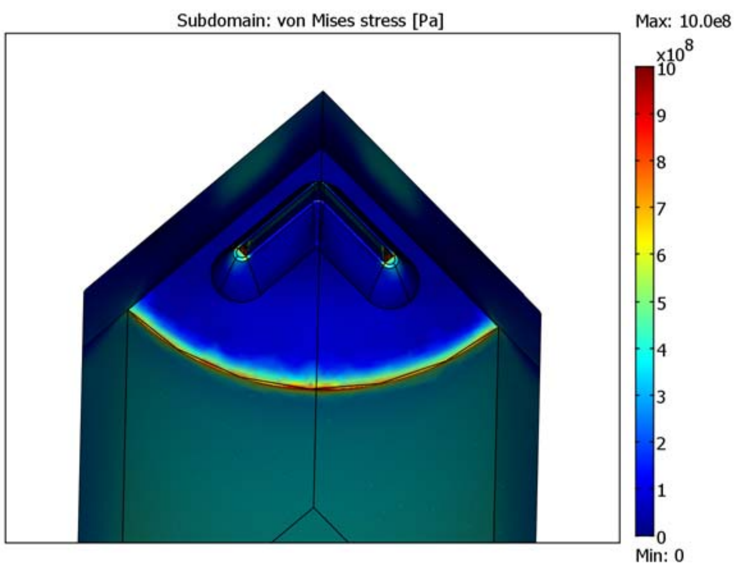


Figure 3-38: Equivalent von Mises distribution in the container.

References

1. M. Schikorra, L. Donati, L. Tomesani, and A.E. Tekkaya, "The Extrusion Benchmark 2007," *Proceedings of the Extrusion Workshop 2007 and 2nd Extrusion Benchmark Conference*, Bologna, Italy, http://diemtech.ing.unibo.it/extrusion07.

2. E.D. Schmitter, "Modelling massive forming processes with thermally coupled fluid dynamics," *Proceedings of the COMSOL Multiphysics User's Conference 2005 Frankfurt*, Frankfurt, Germany.

Model Library path: `Heat_Transfer_Module/Process_and_Manufacturing/aluminum_extrusion_fsi`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1. Double-click the COMSOL Multiphysics icon on your desktop to open the **Model Navigator**.
2. Select **3D** from the **Space dimension** list and click the **Multiphysics** button.
3. From the list of application modes, select **Structural Mechanics Module>Solid, Stress-Strain**, then click **Add**.
4. Select **Chemical Engineering Module>Momentum Transport>Laminar Flow>Non-Newtonian Flow**, then click **Add**.
5. Select **Heat Transfer Module>General Heat Transfer**. Change the **Element** to **Lagrange - Linear**, then click **Add**.
6. Click **OK**.

OPTIONS AND SETTINGS

1. From the **Options** menu, select **Constants**.
2. Enter the following constant names, expressions, and descriptions (the descriptions are optional); when finished, click **OK**.

NAME	EXPRESSION	DESCRIPTION
K_al	210[N/(s*K)]	Thermal conductivity, aluminum
Rho_al	2700[kg/m^3]	Density, aluminum

NAME	EXPRESSION	DESCRIPTION
Cp_al	2.94[N/(mm^2*K)]/Rho_al	Specific heat, aluminum
D_alfe	1[mm]	Thickness, highly conductive layer
Heat_alfe	11[N/(s*mm*K)]	Aluminum-steel heat exchange coefficient
T_billet	460[degC]	Temperature, billet
T_container	450[degC]	Temperature, container
T_ram	380[degC]	Temperature, ram
T_pd1	404[degC]	Initial temperature around thermocouple at point PD1
V_ram	0.5[mm/s]	Velocity, ram
P_init	0[bar]	External reference pressure
Radius	0.073[m]	Radius, ram
T_Air	25[degC]	Ambient temperature
Q_eta	153000[J/mol]	Parameter Q for the generalized Zener Hollomon function
R_eta	8.314[J/(K*mol)]	Ideal gas constant
n_eta	2.976	Parameter n for the generalized Zener-Hollomon function
A_eta	2.39e8[1/s]	Parameter A for the generalized Zener-Hollomon function
Alfa_eta	0.052[1/MPa]	Parameter alpha for the generalized Zener-Hollomon function
H_conv	15	Convective heat exchange coefficient with air
Factor	sqrt(1/3)	Factor for the conversion of the shear rate to COMSOL's definition

GEOMETRY MODELING

Import the model geometry from a CAD file:

1. Choose **File>Import>CAD Data From File**.
2. In the **Look in** list, browse to the folder `models/Structural_Mechanics_Module/Fluid-Structure_Interaction` in your COMSOL Multiphysics installation directory.
3. Select the file `aluminum_extrusion_fsi.mphbin`, then click **Import**.

PHYSICS SETTINGS

Start by defining the physical properties in the different subdomains.

SUBDOMAIN SETTINGS—HEAT TRANSFER

1 From the **Physics** menu, select **Subdomain Settings**.

2 In the **Subdomain selection** list, select Subdomains 1 and 2.

3 Click the **Load** button to open the **Materials/Coefficients Library** dialog box.

4 In the **Search string** edit field, type AISI, then click the **Search** button. From the **Search results** list, select **H11 mod (AISI 610)**, then click **OK**.

5 Change the entry in the $\mathbf{C_p}$ edit field so that it reads 4.63[N/(mm^2*K)]/mat1_rho(T[1/K])[kg/m^3].

6 Select Subdomains 3–5. Set **k** to K_al, ρ to Rho_al, and $\mathbf{C_p}$ to Cp_al.

7 Click the **Convection** tab. Select the **Enable convective heat transfer** check box, then enter the velocity components u2, v2, and w2 in the **u** edit fields. Select the **Viscous heating** check box, then specify the dynamic viscosity η as eta_chns.

8 Click the **Init** tab. Set the initial temperatures according to the following table:

SETTING	SUBDOMAIN 1	SUBDOMAINS 2, 4	SUBDOMAIN 3	SUBDOMAIN 5
$T(t_0)$	T_container	T_pd1	T_billet	T_air

9 Click **OK**.

SUBDOMAIN SETTINGS—STRUCTURAL MECHANICS

1 From the **Multiphysics** menu, select **Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomains 3–5, then clear the **Active in this subdomain** check box to deactivate these subdomains.

4 Select Subdomains 1 and 2. From the **Library material** list, select **H11 mod (AISI610)**.

5 On the **Load** page, select the **Include thermal expansion** check box and set the **Strain temperature** to T for both subdomains. Set the **Strain ref. temperature** for Subdomain 1 to T_container and that for Subdomain 2 to T_pd1.

6 Click **OK**.

SUBDOMAIN SETTINGS—NON-NEWTONIAN FLOW

1 From the **Multiphysics** menu, select **Non-Newtonian Flow (chns)**.

2 From the **Physics** menu, select **Properties**.

3 Set the **Weakly compressible flow** option to **On**, then click **OK**.

4 From the **Physics** menu, select **Subdomain Settings**.

5 Select Subdomains 1 and 2, then clear the **Active in this subdomain** check box.

6 Select Subdomains 3–5. From the **Viscosity model** list, select **User-defined model**. Set ρ to `Rho_al`, η to `Eta_al`, and κ_{dv} to `Eta_al/3`.

7 On the **Init** page, set **P2(t$_0$)** to `P_init`.

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

9 Select **Options>Expressions>Subdomain Expressions**.

10 Create two new expressions for Subdomains 3–5 by entering the following data:

NAME	EXPRESSION
Z_eta	Factor*sr_chns*exp(Q_eta/(R_eta*T))
Eta_al	asinh((Z_eta/A_eta)^(1/n_eta))/ (3*Alfa_eta*Factor*sr_chns+sqrt(eps))

11 Click **OK**.

BOUNDARY CONDITIONS—NON-NEWTONIAN FLOW

1 From the **Physics menu**, select **Boundary Settings**.

2 Select Boundaries 10 and 96, then set the **Boundary type** to **Symmetry boundary**.

3 Select Boundary 11. Set the **Boundary type** to **Inlet** and the **Boundary condition** to **Velocity**. In the $\mathbf{w_0}$ edit field, type `V_ram`.

4 Select Boundaries 36, 37, 41, 45, 47, 62, 68, 69, 73, 79, 85, and 87. Set the **Boundary condition** to **Slip**.

5 Select Boundary 39. Set the **Boundary type** to **Outlet** and the **Boundary condition** to **Normal stress**. In the $\mathbf{f_0}$ edit field, type `P_init`.

6 Click **OK**.

BOUNDARY CONDITIONS—STRUCTURAL MECHANICS

1 From the **Multiphysics** menu, select **Solid, Stress-Strain (smsld)**.

2 From the **Physics** menu, select **Boundary Settings**.

3 Select Boundaries 2, 5, 7, and 8, then set the **Constraint condition** to **Roller**.

4 Select Boundaries 1, 4, 94, and 95, then set the **Constraint condition** to **Symmetry plane**.

5 Select Boundaries 9, 12, 15, 16, 21, 22, 26, 30, 31, 32, 33, 34, 35, 40, 44, 46, 50–52, 56, 64–66, 67, 72, 74–78, 84, 86, 88, and 91. On the **Load** page, set the **Coordinate system** to **Tangent and normal coord. sys.** and **Fn** to `p2`.

6 Select Boundary 61, then set the **Coordinate system** to **Tangent and normal coord. sys.** and **Fn** to `-p2`.

7 Click **OK**.

BOUNDARY CONDITIONS—HEAT TRANSFER

1 From the **Multiphysics** menu, select **General Heat Transfer (htgh)**.

2 From the **Physics** menu, select **Boundary Settings**.

3 Select Boundaries 2, 5, 7, and 8, then set the **Boundary condition** to **Temperature**. In the $\mathbf{T_0}$ edit field, type `T_container`.

4 Select Boundary 11, then set the **Boundary condition** to **Heat flux**. Set **h** to `Heat_Alfe` and $\mathbf{T_{inf}}$ to `T_ram`.

5 Select Boundaries 36, 37, 41, 45, 47, 62, 68, 69, 73, 79, 85, and 87, then set the **Boundary condition** to **Heat flux**. Set **h** to `H_conv` and $\mathbf{T_{inf}}$ to `T_air`.

6 Select Boundary 39, then set the **Boundary condition** to **Convective flux**.

7 Select the **Interior boundaries** check box, then select Boundaries 9, 12, 15, 16, 21, 22, 26, 30–35, 40, 44, 46, 50–52, 56, 61, 64–67, 72, 74–78, 84, 86, 88, and 91. On the **Highly Conductive Layer** page, select the **Enable heat transfer in highly conductive layer** check box. Set $\mathbf{k_s}$ to `Heat_Alfe*D_alfe` and $\mathbf{d_s}$ to `D_alfe`.

8 Click **OK**.

MESH GENERATION

1 Choose **Mesh>Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list, select **Coarser**.

3 Click the **Subdomain** tab. Select Subdomains 1–3, then set the **Maximum element size** to `0.0085`. Select subdomain 4, then set the **Maximum element size** to `0.0025`.

4 Click the **Boundary** tab. Select Boundaries 2, 5, and 8, then set the **Maximum element size** to `0.0085`.

5 Select Boundaries 13–16, 21, 22, 26, 29–35, 38, 40, 44, 46, 50–52, 56, 61, 64–67, 72, 74–78, 84, 86, 88, and 91, then set the **Maximum element size** to `0.002`.

6 Select Boundaries 18–20, 23–25, 27, 28, 42, 43, 48, 49, 54, 55, 57–60, 63, 70, 71, 80–83, 89, 90, 92, and 93, then set the **Maximum element size** to `0.0025`.

7 Click **Remesh**.

8 Click **OK**.

9 Select Subdomain 5 and from the interactive mesh column delete the mesh on this subdomain. Select **Mesh>Swept Mesh Parameters** and set the **Number of element layers** to 24. Click **Mesh Selected**.

10 Click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 From the **Linear system solver** list, select **PARDISO**. Click the **Settings** button.

3 In the **Linear System Solver Settings** dialog box, set **Check tolerances** to **Off**. Click **OK**.

4 Click the **Stationary** tab. Select the **Manual tuning of damping parameters** check box, then set the **Minimum damping factor** to `1e-8`.

5 Click **OK** to close the **Solver Parameters** dialog box.

6 Click the **Solver Manager** button on the Main toolbar.

7 On the **Solve For** page, select **General Heat Transfer (htgh)** only in the **Solve for variables** list.

8 Click the **Solve** button.

It is important to obtain a good initial guess for the velocity field. To do this, temporarily set the properties of the fluid-dynamics part to those for incompressible Navier-Stokes flow:

9 Choose first **Multiphysics>Non-Newtonian Flow (chns)** and then **Physics>Properties**. Set both the **Weakly compressible flow** and **Non-Newtonian Flow** options to **Off**, then click **OK**.

10 Return to the **Solver Manager**. On the **Solve For** page, select **Incompressible Navier-Stokes (chns)** in the **Solve for variables** list.

11 On the **Initial Value** page, select as the **Initial value** the **Current solution**.

12 Click the **Solve** button.

13 On the **Solve For** page, select **General Heat Transfer (htgh)** in the **Solve for variables** list.

14 Click the **Solve** button.

15 Now reset the fluid-dynamics properties to those for Non-Newtonian flow. Choose **Physics>Properties**. Set both the **Weakly compressible flow** and **Non-Newtonian Flow** options to **On**, then click **OK**.

16 Return to the **Solver Manager** and on the **Solve For** page, select **Non-Newtonian Flow (chns)** and **General Heat Transfer (htgh)** from the **Solve for variables** list.

17 Click the **Solve** button.

Now turn to the structural mechanics analysis.

18 From the **Solve for variables** list, select **Solid, Stress-Strain (smsld)** only.

19 Click **OK** to close the **Solver Manager**.

20 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

To generate the plot shown in Figure 3-38 on page 268, follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar to open the **Plot Parameters** dialog box.

2 In the **Plot type** area on the **General** page, clear the **Slice** check box and select the **Subdomain** check box.

3 On the **Subdomain** page, select **Solid, Stress-Strain (smsld)>von Mises stress** from the **Predefined quantities** list.

4 Click the **Range** button to change the plot range.

5 In the **Color Range** dialog box, clear the **Auto** check box and enter `0` in the **Min** edit field and `1e9` in the **Max** edit field. Click **OK**.

6 Click **Apply** to show the plot.

7 Rotate the geometry to find the interesting stress concentrations on the inside.

With the following steps you can reproduce the temperature plot in Figure 3-36:

1 On the **General** page, clear the **Subdomain** check box and select the **Slice** check box.

2 Click the **Slice** tab. From the **Predefined quantities** list, select **General Heat Transfer (htgh)>Temperature**.

3 From the **Unit** list, select **degC**.

4 Click the **Vector with coordinates** buttons for the **x levels** and **y levels**. For both directions, type `-0.01475+0.0015` in the associated edit field.

5 From the **Options** menu, select **Suppress Subdomains**.

6 From the **Subdomain selection** list, select Subdomains 1 and 2, then click **OK**.

7 Click **Apply** to plot the results.

8 Rotate the geometry to view the temperature distribution.

Finally, to visualize the velocity field, as in Figure 3-37, do as follows:

1 In the **Plot Parameters** dialog box, click the **Slice** tab.

2 From the **Predefined quantities** list, select **Non-Newtonian Flow (chns)>Velocity field**.

3 Click the **x levels** and **y levels** option buttons, then type `0` in the corresponding **Number of levels** edit fields.

4 Click the **Vector with coordinates** button for the **z levels**. In the associated edit field, type `0.0151`.

5 On the **Streamline** page, select the **Streamline plot** check box.

6 From the **Predefined quantities** list, select **Non-Newtonian Flow (chns)>Velocity field**.

7 From the **Streamline plot type** list, select **Uniform density**. Set the **Separating distance** to `0.035`.

8 Click the **Line Color** tab. Click first the **Use expression** button and then the **Color Expression** button. In the **Streamline Color Expression** dialog box, select **Non-Newtonian Flow (chns)>Velocity field** from the **Predefined quantities** list. Because (as it turns out) the plot range effectively coincides with that for the slice plot, you can turn off the color scale for the streamlines. Thus, clear the **Color scale** check box, then click **OK** to close the **Streamline Color Expression** dialog box.

9 Set the **Line type** to **Tube**, then click the **Tube Radius** button.

10 In the **Tube Radius Parameters** dialog box, clear the **Auto** check box. Set the **Radius scale factor** to `0.2`, then click **OK**.

11 Click **OK** to close the **Plot Parameters** dialog box and generate the plot.

12 Rotate the geometry and zoom in around the extruding part.

4

Medical Technology Models

This chapter includes two models that describe heat transfer in medical applications using the bioheat equation.

Microwave Cancer Therapy

Introduction

Electromagnetic heating appears in a wide range of engineering problems and is ideally suited for modeling in COMSOL Multiphysics because of its multiphysics capabilities. This example comes from the area of hyperthermic oncology and it models the electromagnetic field coupled to the bioheat equation. The modeling issues and techniques are generally applicable to any problem involving electromagnetic heating.

In hyperthermic oncology, cancer is treated by applying localized heating to the tumor tissue, often in combination with chemotherapy or radiotherapy. Some of the challenges associated with the selective heating of deep-seated tumors without damaging surrounding tissue are:

- Control of heating power and spatial distribution
- Design and placement of temperature sensors

Among possible heating techniques, RF and microwave heating have attracted much attention from clinical researchers. Microwave coagulation therapy is one such technique where a thin microwave antenna is inserted into the tumor. The microwaves heat up the tumor, producing a coagulated region where the cancer cells are killed.

This model computes the temperature field, the radiation field, and the specific absorption rate (SAR)—defined as the ratio of absorbed heat power and tissue density—in liver tissue when using a thin coaxial slot antenna for microwave coagulation therapy. It closely follows the analysis found in Ref. 1. It computes the temperature distribution in the tissue using the bioheat equation.

Model Definition

Figure 4-1 shows the antenna geometry. It consists of a thin coaxial cable with a ring-shaped slot measuring 1 mm cut on the outer conductor 5 mm from the short-circuited tip. For hygienic purposes, the antenna is enclosed in a sleeve (catheter) made of PTFE (polytetrafluoroethylene). The following tables give the relevant geometrical dimensions and material data. The antenna operates at 2.45 GHz, a frequency widely used in microwave coagulation therapy.

TABLE 4-1: DIMENSIONS OF THE COAXIAL SLOT ANTENNA.

PROPERTY	VALUE
Diameter of the central conductor	0.29 mm
Inner diameter of the outer conductor	0.94 mm
Outer diameter of the outer conductor	1.19 mm
Diameter of catheter	1.79 mm

TABLE 4-2: MATERIAL PROPERTIES.

PROPERTY	INNER DIELECTRIC OF COAXIAL CABLE	CATHETER	LIVER TISSUE
Relative permittivity	2.03	2.60	43.03
Conductivity			1.69 S/m

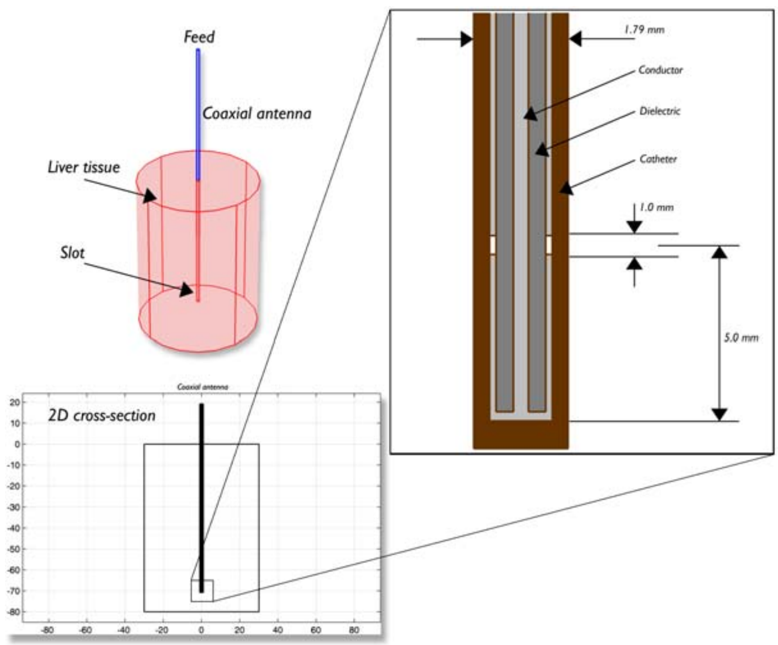


Figure 4-1: Antenna geometry for microwave coagulation therapy. A coaxial cable with a ring-shaped slot cut on the outer conductor is short-circuited at the tip. A plastic catheter surrounds the antenna.

The model takes advantage of the problem's rotational symmetry, which allows modeling in 2D using cylindrical coordinates as indicated in Figure 4-2. When modeling in 2D, you can select a fine mesh and achieve excellent accuracy. The model uses a frequency-domain problem formulation with the complex-valued azimuthal component of the magnetic field as the unknown.

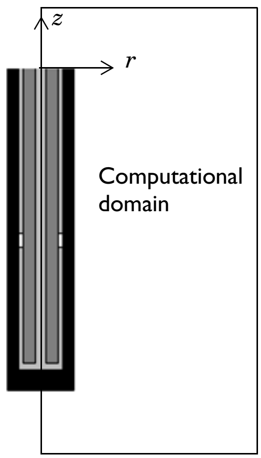


Figure 4-2: The computational domain appears as a rectangle in the rz-plane.

The radial and axial extent of the computational domain is in reality larger than indicated in Figure 4-2. This problem does not model the interior of the metallic conductors, and it models metallic parts using boundary conditions, setting the tangential component of the electric field to zero.

DOMAIN AND BOUNDARY EQUATIONS—ELECTROMAGNETICS

An electromagnetic wave propagating in a coaxial cable is characterized by transverse electromagnetic fields (TEM). Assuming time-harmonic fields with complex amplitudes containing the phase information, the appropriate equations are

$$\mathbf{E} = \mathbf{e}_r \frac{C}{r} e^{j(\omega t - kz)}$$

$$\mathbf{H} = \mathbf{e}_\varphi \frac{C}{rZ} e^{j(\omega t - kz)}$$

$$\mathbf{P}_{\mathrm{av}} = \int_{r_{\mathrm{inner}}}^{r_{\mathrm{outer}}} \mathrm{Re}\left(\frac{1}{2}\mathbf{E} \times \mathbf{H}^*\right) 2\pi r\,dr = \mathbf{e}_z \pi \frac{C^2}{Z} \ln\left(\frac{r_{\mathrm{outer}}}{r_{\mathrm{inner}}}\right)$$

where z is the direction of propagation, and r, φ, and z are cylindrical coordinates centered on the axis of the coaxial cable. $\mathbf{P}_{av}$ is the time-averaged power flow in the cable, Z is the wave impedance in the dielectric of the cable, while r_{inner} and r_{outer} are the dielectric's inner and outer radii, respectively. Further, ω denotes the angular frequency. The propagation constant, k, relates to the wavelength in the medium, λ, as

$$k = \frac{2\pi}{\lambda}.$$

In the tissue, the electric field also has a finite axial component whereas the magnetic field is purely in the azimuthal direction. Thus, you can model the antenna using an axisymmetric transverse magnetic (TM) formulation. The wave equation then becomes scalar in H_φ:

$$\nabla \times \left(\left(\varepsilon_r - \frac{j\sigma}{\omega\varepsilon_0}\right)^{-1} \nabla \times H_\varphi\right) - \mu_r k_0^{\ 2} H_\varphi = 0.$$

The boundary conditions for the metallic surfaces are

$$\mathbf{n} \times \mathbf{E} = 0.$$

The feed point is modeled using a port boundary condition with a power level set to 10 W. This is essentially a first-order low-reflecting boundary condition with an input field $H_{\varphi 0}$:

$$\mathbf{n} \times \sqrt{\varepsilon}\mathbf{E} - \sqrt{\mu} H_\varphi = -2\sqrt{\mu} H_{\varphi 0}$$

where

$$H_{\varphi 0} = \frac{\sqrt{\dfrac{\mathbf{P}_{av} Z}{\pi r \ln\left(\dfrac{r_{outer}}{r_{inner}}\right)}}}{r}$$

for an input power of $\mathbf{P}_{av}$ deduced from the time-average power flow.

The antenna radiates into the tissue where a damped wave propagates. Because you can discretize only a finite region, you must truncate the geometry some distance from the antenna using a similar absorbing boundary condition without excitation. Apply this boundary condition to all exterior boundaries. Finally, apply a symmetry boundary condition for boundaries at $r = 0$.

DOMAIN AND BOUNDARY EQUATIONS—HEAT TRANSFER

The bioheat equation describes the stationary heat transfer problem as

$$\nabla \cdot (-k\nabla T) = \rho_b C_b \omega_b (T_b - T) + Q_{met} + Q_{ext}$$

where k is the liver's thermal conductivity (W/(m·K)), ρ_b represents the blood density (kg/m^3), C_b is the blood's specific heat capacity (J/(kg·K)), and ω_b denotes the blood perfusion rate (1/s). Further, Q_{met} is the heat source from metabolism, and Q_{ext} is an external heat source, both measured in W/m^3.

This model neglects the heat source from metabolism. The external heat source is equal to the resistive heat generated by the electromagnetic field:

$$Q_{ext} = \frac{1}{2}\mathrm{Re}[(\sigma - j\omega\varepsilon)\mathbf{E} \cdot \mathbf{E}^*] .$$

The model assumes that the blood perfusion rate is $\omega_b = 0.0036\ \mathrm{s}^{-1}$, and that the blood enters the liver at the body temperature $T_b = 37$ °C and is heated to a temperature, T. The blood's specific heat capacity is $C_b = 3639$ J/(kg·K).

For a more realistic model, you might consider letting ω_b be a function of the temperature. At least for external body parts such as hands and feet, it is evident that a temperature increase results in an increased blood flow.

This example models the heat-transfer problem only in the liver domain. Where this domain is truncated, it uses insulation, that is

$$\mathbf{n} \cdot \nabla T = 0 .$$

Results and Discussion

Figure 4-3 shows the resulting steady-state temperature distribution in the liver tissue for an input microwave power of 10 W. The temperature is highest near the antenna. It then decreases with distance from the antenna and reaches 37 °C closer to the outer boundaries of the computational domain. The perfusion of relatively cold blood seems to limit the extent of the area that is heated.

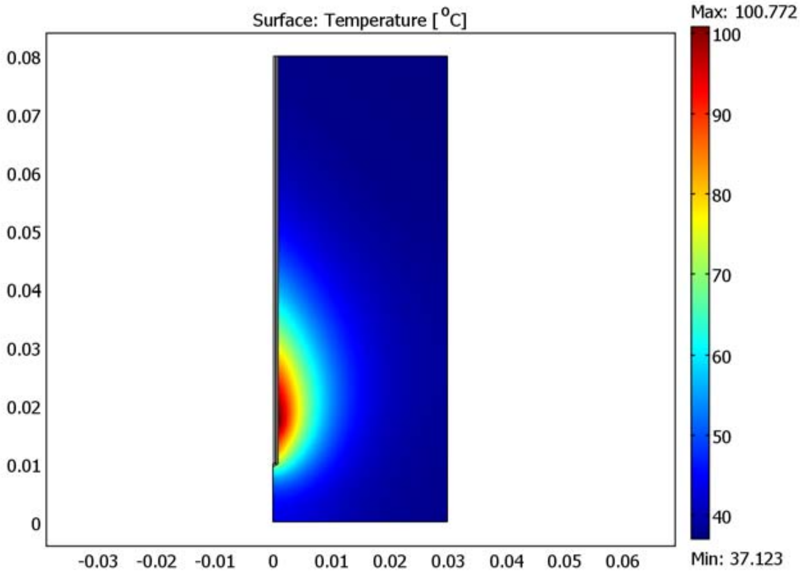


Figure 4-3: Temperature in the liver tissue.

Figure 4-4 shows the distribution of the microwave heat source. Clearly the temperature field follows the heat-source distribution quite well. That is, near the antenna the heat source is strong, which leads to high temperatures, while far from the antenna, the heat source is weaker and the blood manages to keep the tissue at normal body temperature.

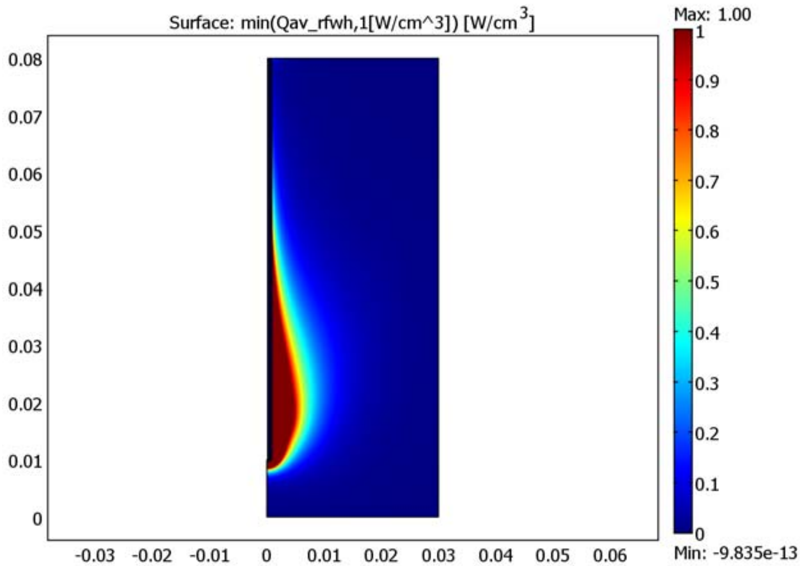


Figure 4-4: The computed microwave heat-source density takes on its highest values near the tip and the slot. The scale is cut off at 1 W/cm^3.

Figure 4-5 plots the specific absorption rate (SAR) along a line parallel to the antenna and at a distance of 2.5 mm from the antenna axis normalized by its maximal value along the line. The results are in good agreement with those found in Ref. 1.

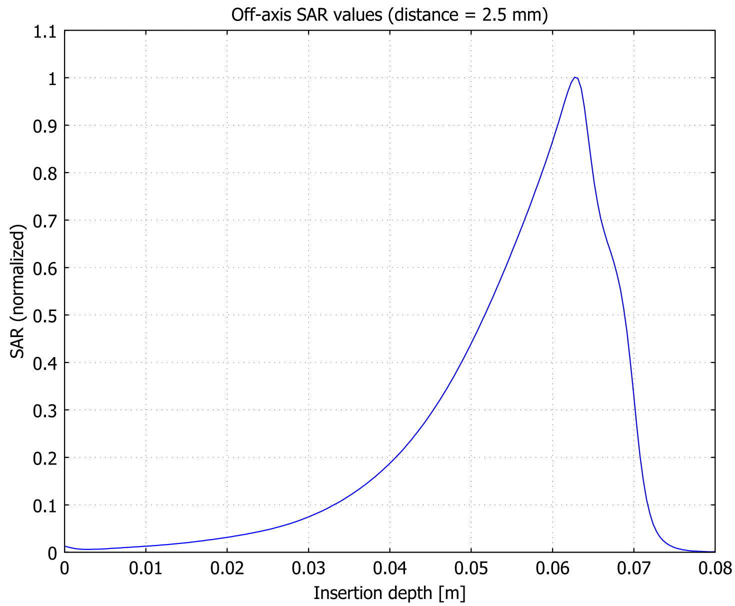


Figure 4-5: Normalized SAR value along a line parallel to the antenna and at a distance 2.5 mm from the antenna axis. The tip of the antenna is located at 70 mm, and the slot is at 65 mm.

Reference

1. K. Saito, T. Taniguchi, H. Yoshimura, and K. Ito, "Estimation of SAR Distribution of a Tip-Split Array Applicator for Microwave Coagulation Therapy Using the Finite Element Method," *IEICE Trans. Electronics*, vol. E84-C, 7, pp. 948–954, July 2001.

Modeling in COMSOL Multiphysics

The COMSOL Multiphysics implementation is straightforward. Drawing the geometry is best done creating rectangles and setting their dimensions directly from the **Draw** menu. The scale differences together with the strong radial dependence of the electromagnetic fields make some manual adjustment of the mesh parameters necessary. In addition, 4th-order elements for the electromagnetic problem and a dense mesh in the dielectric result in well-resolved fields. The model computes the solutions for both the electromagnetic problem and the heat transfer problem in parallel. This takes into account the coupling of the resistive heating from the electromagnetic solution into the bioheat equation. In principle, however, you could solve the two problems in sequence because there is only a 1-way coupling from the electromagnetic problem to the bioheat problem.

Model Library path: `Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy`

Note: This model requires the RF Module and the Heat Transfer Module.

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**. In the **Space dimension** list select **Axial symmetry 2D**.

2 In the list of application modes select **Heat Transfer Module>Bioheat Equation>Steady-state analysis**.

3 Click the **Multiphysics** button, then click the **Add** button.

4 In the list of application modes select **RF Module>Electromagnetic Waves>TM Waves>Harmonic propagation**.

5 In the **Element** list select **Lagrange - Quartic**.

6 Click **Add**, then click **OK**.

OPTIONS AND SETTINGS

From the **Options** menu select **Constants**. Enter the following names and expressions; when done, click **OK**.

NAME	EXPRESSION	EXPRESSION
k_liver	0.56[W/(kg*K)]	Thermal conductivity, liver
rho_blood	1000[kg/m^3]	Density, blood
C_blood	3639[J/(kg*K)]	Specific heat, blood
omega_blood	3.6e-3[1/s]	Blood perfusion rate

NAME	EXPRESSION	EXPRESSION
T_blood	37[degC]	Blood temperature
P_in	10[W]	Input microwave power
nu	2.45[GHz]	Microwave frequency
eps_diel	2.03	Relative permittivity, dielectric
eps_cat	2.6	Relative permittivity, catheter
eps_liver	43.03	Relative permittivity, liver
sig_liver	1.69[S/m]	Electric conductivity

GEOMETRY MODELING

1 Create two rectangles. Select the menu item **Draw>Specify Objects>Rectangle**, then enter the following settings; when done with each one, click **OK**.

WIDTH	HEIGHT	BASE CORNER R	BASE CORNER Z
0.595e-3	0.01	0	0
29.405e-3	0.08	0.595e-3	0

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

3 From the Draw menu open the **Create Composite Object** dialog box. Clear the **Keep interior boundaries** check box. In the **Object selection** box select both rectangles, then click the **Union** button. Click **OK**.

4 Following the procedure in Step 1, specify two more rectangles with the following properties:

WIDTH	HEIGHT	BASE CORNER R	BASE CORNER Z
0.125e-3	1e-3	0.47e-3	0.0155
0.335e-3	0.0699	0.135e-3	0.0101

5 Add a line to the geometry. Select the menu item **Draw>Specify Objects>Line.** In the **r** edit field enter the coordinates `0 8.95e-4 8.95e-4`, and in the **z** edit field enter the coordinates `9.5e-3 0.01 0.08`. Click **OK**.

PHYSICS SETTINGS

Subdomain Settings—Bioheat Equation

1 From the **Multiphysics** menu select **1 Bioheat Equation (htbh)**.

2 From the **Physics** menu select **Subdomain Settings**.

3 Select Subdomains 2, 3, and 4, then clear the **Active in this domain** check box.

4 Select Subdomain 1, then enter the following settings; when done, click **OK**.

PROPERTY	VALUE
k (isotropic)	`k_liver`
ρ_b	`rho_blood`
C_b	`C_blood`
ω_b	`omega_blood`
T_b	`T_blood`
Q_{met}	`0`
Q_{ext}	`Qav_rfwh`

Boundary Conditions—Bioheat Equation

1 From the **Physics** menu select **Boundary Settings**.

2 Select all the exterior boundaries (get them by pressing Ctrl+A, and note that the following step ignores the interior boundaries).

3 In the **Boundary condition** list select **Thermal insulation**, then click **OK**.

Note: Because the model neglects metabolic heat generation you set Q_{met} to `0`. The variable `Qav_rfwh` is a subdomain expression for the resistive heating provided by the TM Waves application mode.

Scalar Variables—TM Waves

1 From the **Multiphysics** menu select **2 TM Waves (rfwh)**.

2 From the **Physics** menu select **Scalar Variables** to open the **Application Scalar Variables** dialog box.

3 Find the variable `nu_rfwh` and set its value to `nu`, then click **OK**.

Boundary Conditions—TM Waves

1 From the **Physics** menu select **Boundary Settings**.

2 Specify boundary settings according to the following table (to enter the port settings for Boundary 8 go to the **Port** page); when finished, click **OK**.

SETTINGS	BOUNDARIES 1, 3	BOUNDARIES 2, 14, 18, 20, 21	BOUNDARY 8
Boundary condition	Axial symmetry	Scattering boundary condition	Port
Wave excitation at this port			selected
P_{in}			`P_in`
Mode specification			Coaxial
Wave type		Spherical wave	

For the (exterior) boundaries not mentioned in the table, the default condition (perfect electric conductor) applies.

Subdomain Settings—TM Waves

1 From the **Physics** menu select **Subdomain Settings**.

2 Enter the following settings; when finished, click **OK**.

SETTINGS	SUBDOMAIN 1	SUBDOMAIN 2	SUBDOMAIN 3	SUBDOMAIN 4
ε_r (isotropic)	`eps_liver`	`eps_cat`	`eps_diel`	1
σ (isotropic)	`sig_liver`	0	0	0
μ_r	1	1	1	1

MESH GENERATION

1 From the **Mesh** menu open the **Free Mesh Parameters** dialog box.

2 Go to the **Global** page, click the **Custom mesh size** button and in the **Maximum element size** edit field type `3e-3`.

3 Go to the **Subdomain** page and select Subdomain 3. In the **Maximum element size** edit field type `1.5e-4`.

4 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot shows the temperature field. To change the unit to degrees Celsius, reproducing the plot in Figure 4-3, do as follows:

1 Click the **Plot Parameters** button on the Main toolbar.
2 Click the **Surface** tab. From the **Unit** list select **°C**, then click **Apply**.

The following steps describe how to visualize the resistive heating of the tissue:

1 In the **Predefined quantities** list select **TM Waves (rfwh)>Resistive heating, time average**. In the **Unit** edit field type `W/cm^3`, then click **Apply**.

   Heating decreases rapidly in the liver tissue, resulting in an almost uniformly blue plot. To get a better feeling for the heating at a distance from the antenna, do as follows:

2 In the **Expression** edit field type `min(Qav_rfwh,1[W/cm^3])`, then click **OK**.

   In the resulting plot, which reproduces that in Figure 4-4, the region around the antenna in which the time-averaged resistive heating exceeds 1 W/cm^3 has a uniform, deep red color. Outside this region, you can read off the heating distribution from the color scale on the right.

To compute the total heating power deposited in the liver, follow these steps:

1 From the **Postprocessing** menu open the **Subdomain Integration** dialog box.
2 Select Subdomain 1. From the **Predefined quantities** list select **TM Waves (rfwh)>Resistive heating, time average**.
3 Select the **Compute volume integral (for axisymmetric modes)** check box. Click **OK**.

   The result appears in the message log at the bottom of the user interface. The value of approximately 9.37 W indicates that the tissue absorbs most of the 10 W input power at stationary conditions.

These steps reproduce the plot in Figure 4-5, displaying the normalized SAR value:

1 From the **Postprocessing** menu open the **Cross-Section Plot Parameters** dialog box.
2 On the **Line/Extrusion** page, type `Qav_rfwh/3.01[W/cm^3]` in the **Expression** edit field. In both the **r0** and **r1** edit fields type `2.5e-3`; in the **z0** edit field type `0.08`; and in the **z1** edit field type `0`.
3 Click the **General** tab, then click the **Title/Axis** button. In the **Title/Axis Settings** dialog box, select the option button next to the **Title** edit field, then enter the title `Off-axis SAR values (distance = 2.5 mm)`.
4 In a similar way, enter the first axis label `Insertion depth [m]` and the second axis label `SAR (normalized)`, then click **OK** to close the **Title/Axis Settings** dialog box.
5 Click **OK** to generate the plot.

Tumor Removal

Introduction

One method for removing cancerous tumors from healthy tissue is to heat the malignant tissue to a critical temperature that kills the cancer cells. This example accomplishes the localized heating by inserting a four-armed electric probe through which an electric current runs. Equations for the electric field for this case appear in the Conductive Media DC application mode, and this example couples them to the bioheat equation, which models the temperature field in the tissue. The heat source resulting from the electric field is also known as *resistive heating* or *Joule heating*. The original model comes from S. Tungjitkusolmun and others (Ref. 1), but we have made some simplifications. For instance, while the original uses RF heating (with AC currents), the COMSOL Multiphysics model approximates the energy with DC currents.

This medical procedure removes the tumorous tissue by heating it above 45 °C to 50 °C. Doing so requires a local heat source, which physicians create by inserting a small electric probe. The probe is made of a trocar (the main rod) and four electrode arms as shown in Figure 4-6. The trocar is electrically insulated except near the electrode arms.

An electric current through the probe creates an electric field in the tissue. The field is strongest in the immediate vicinity of the probe and generates resistive heating, which dominates around the probe's electrode arms because of the strong electric field.

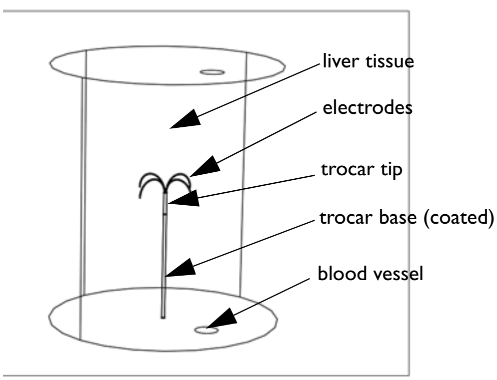


Figure 4-6: Cylindrical modeling domain with the four-armed electric probe in the middle, which is located next to a large blood vessel.

Model Definition

This model uses the Bioheat Equation and the Conductive Media DC application modes to implement a transient analysis.

The standard temperature unit in COMSOL Multiphysics is kelvin (K). This model uses the Celsius temperature scale, which is more convenient for models involving the Bioheat Equation.

The model approximates the body tissue with a large cylinder and assumes that its boundary temperature remains at 37 °C during the entire procedure. The tumor is located near the center of the cylinder and has the same thermal properties as the surrounding tissue. The model locates the probe along the cylinder's center line such that its electrodes span the region where the tumor is located. The geometry also includes a large blood vessel.

The bioheat equation governs heat transfer in the tissue

$$\delta_{ts}\rho C\frac{\partial T}{\partial t} + \nabla \cdot (-k\nabla T) = \rho_b C_b \omega_b (T_b - T) + Q_{met} + Q_{ext}$$

where δ_{ts} is a time-scaling coefficient; ρ is the tissue density (kg/m^3); C is the tissue's specific heat (J/(kg·K)); and k is its thermal conductivity (W/(m·K)). On the right side of the equality, ρ_b gives the blood's density (kg/m^3); C_b is the blood's specific heat (J/(kg·K)); ω_b is its perfusion rate (1/s); T_b is the arterial blood temperature (°C); while Q_{met} and Q_{ext} are the heat sources from metabolism and spatial heating, respectively (W/m^3).

In this model, the bioheat equation also models heat transfer in various parts of the probe with the appropriate values for the specific heat, C (J/(kg·K)), and thermal conductivity, k (W/(m·K)). For these parts, all terms on the right-hand side are zero.

The model next sets the boundary conditions at the outer boundaries of the cylinder and at the walls of the blood vessel to a temperature of 37 °C. Assume heat flux continuity on all other boundaries.

The initial temperature equals 37 °C in all domains.

The governing equation for the Conductive Media DC application mode is

$$-\nabla \cdot (\sigma\nabla V - \mathbf{J}^e) = Q_j$$

where V is the potential (V), σ the electric conductivity (S/m), $\mathbf{J}^e$ an externally generated current density (A/m^2), Q_j the current source (A/m^3).

In this model both $\mathbf{J}^e$ and Q_j are zero. The governing equation therefore simplifies into:

$$-\nabla \cdot (\sigma \nabla V) = 0 .$$

The boundary conditions at the cylinder's outer boundaries is ground (0 V potential). At the electrode boundaries the potential equals 22 V. Assume continuity for all other boundaries.

The boundary conditions for the Conductive Media DC application mode are:

$V = 0$	on the cylinder wall
$V = V_0$	on the electrode surfaces
$\mathbf{n} \cdot (J_1 - J_2) = 0$	on all other boundaries

The boundary conditions for the bioheat equation are:

$T = T_b$	on the cylinder wall and blood-vessel wall
$\mathbf{n} \cdot (k_1 \nabla T_1 - k_2 \nabla T_2) = 0$	on all interior boundaries

The model solves the above equations with the given boundary conditions to obtain the temperature field as a function time.

Results and Discussion

The model shows how the temperature increases with time in the tissue around the electrode.

The slice plot in Figure 4-7 illustrates the temperature field 60 seconds after starting the procedure.

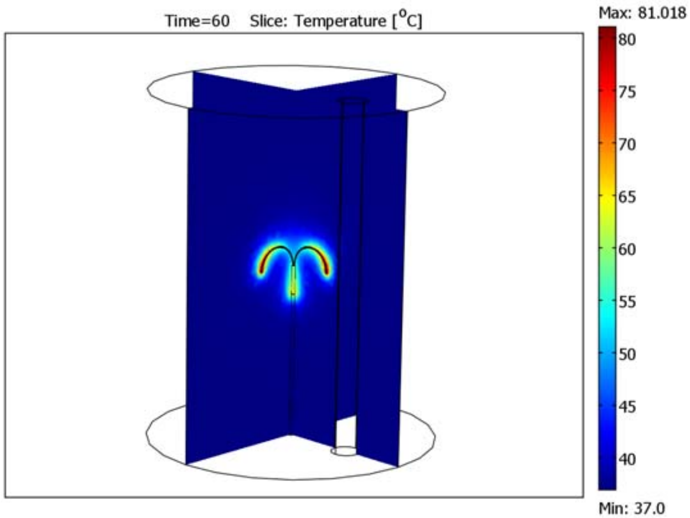


Figure 4-7: Temperature field at time = 60 seconds.

Figure 4-8 shows the temperature at the tip of one of the electrode arms. The temperature rises quickly until it reaches a steady-state temperature of about 90 °C.

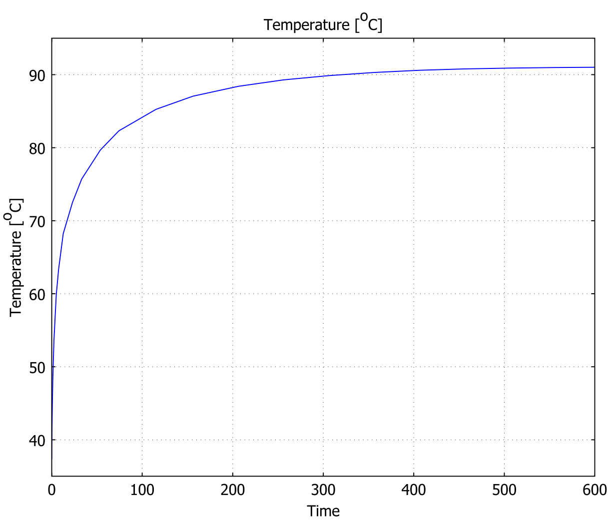


Figure 4-8: Temperature versus time at the tip of one of the electrode arms.

It is also interesting to visualize the region where cancer cells die, that is, where the temperature has reached at least 50 °C. You can visualize this area with an isosurface for that temperature; Figure 4-9 shows one after 8 minutes.

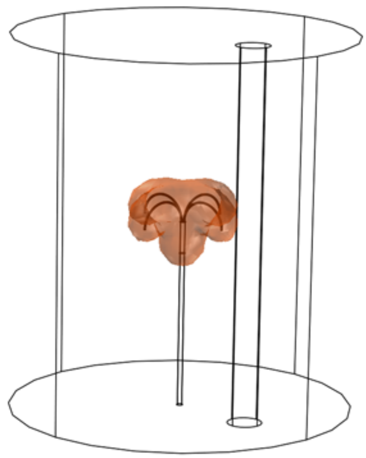


Figure 4-9: Visualization of the region that has reached 50 °C after 8 minutes.

Reference

1. S. Tungjitkusolmun, S. Tyler Staelin, D. Haemmerich, J.Z. Tsai, H. Cao, J.G. Webster, F.T. Lee, Jr., D.M. Mahvi, and V.R. Vorperian, "Three-Dimensional Finite Element Analyses for Radio-Frequency Hepatic Tumor Ablation," *IEEE Transactions on Biomedical Engineering*, vol. 49, no. 1, 2002.

Model Library path: `Heat_Transfer_Module/Medical_Technology/tumor_ablation`

Modeling Using the Graphical User Interface

MODEL NAVIGATOR

1 Open the **Model Navigator**. On the **New** page, select **3D** in the **Space dimension** list.

2 Go to the **Heat Transfer Module** menu and select **Bioheat Equation>Transient analysis**.

3 Click the **Multiphysics** button and add the application mode to the model by clicking the **Add** button.

4 Similarly add the **Conductive Media DC** application mode, from the **COMSOL Multiphysics>Electromagnetics** menu.

5 Click **OK**.

OPTIONS AND SETTINGS

1 From the **Options** menu, choose **Constants**.

2 Define the following constants (the descriptions are optional):

NAME	EXPRESSION	DESCRIPTION
rho_e	6450[kg/m^3]	Density, electrodes
rho_t	21500[kg/m^3]	Density, trocar tip
rho_l	1060[kg/m^3]	Density, liver tissue
rho_b	1000[kg/m^3]	Density, blood
rho_c	70[kg/m^3]	Density, trocar base
c_e	840[J/(kg*K)]	Heat capacity, electrodes
c_t	132[J/(kg*K)]	Heat capacity, trocar tip
c_l	3600[J/(kg*K)]	Heat capacity, liver tissue
c_b	4180[J/(kg*K)]	Heat capacity, blood
c_c	1045[J/(kg*K)]	Heat capacity, trocar base
k_e	18[W/(m*K)]	Thermal conductivity, electrodes
k_t	71[W/(m*K)]	Thermal conductivity, trocar tip
k_l	0.512[W/(m*K)]	Thermal conductivity, liver tissue
k_b	0.543[W/(m*K)]	Thermal conductivity, blood
k_c	0.026[W/(m*K)]	Thermal conductivity, trocar base
sigma_e	1e8[S/m]	Electric conductivity, electrodes
sigma_t	4e6[S/m]	Electric conductivity, trocar tip
sigma_l	0.333[S/m]	Electric conductivity, liver tissue
sigma_b	0.667[S/m]	Electric conductivity, blood
sigma_c	1e-5[S/m]	Electric conductivity, trocar base
omega_b	6.4e-3[1/s]	Blood perfusion rate
T_b	37[degC]	Arterial blood temperature
T0	37[degC]	Initial and boundary temperature
V0	22[V]	Electric voltage

3 Click **OK**.

GEOMETRY MODELING

1 Go to the **Draw** menu and select **Work-Plane Settings**.
2 Create an **x-y plane** at the **z** coordinate `0.06`.
3 Click **OK**.
4 Press the Shift key and click the **Ellipse/Circle (Centered)** button.
5 In the dialog box that appears, enter the following circle properties:

OBJECT DIMENSIONS	EXPRESSION
Radius	9.144e-4
Base	Center
x	0
y	0

6 Click **OK**.
7 Click the **Zoom Extents** button on the Main toolbar.
8 Repeat Steps 4–6 to create five additional circles, for smaller ones and a large one. The properties of each circle appear in the following five tables:

OBJECT DIMENSIONS	EXPRESSION
Radius	2.667e-4
Base	Center
x	-5e-4
y	0

OBJECT DIMENSIONS	EXPRESSION
Radius	2.667e-4
Base	Center
x	5e-4
y	0

OBJECT DIMENSIONS	EXPRESSION
Radius	2.667e-4
Base	Center
x	0
y	-5e-4

OBJECT DIMENSIONS	EXPRESSION
Radius	2.667e-4
Base	Center
x	0
y	5e-4

OBJECT DIMENSIONS	EXPRESSION
Radius	5e-3
Base	Center
x	2.6e-2
y	0

9 Click the **Zoom Extents** button on the Main toolbar.

This completes the geometry needed in the 2D work plane, and the result appears in Figure 4-10.

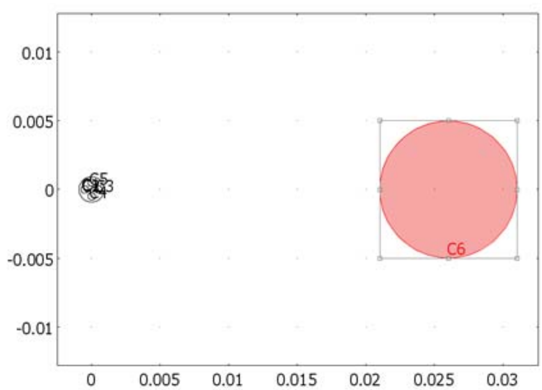


Figure 4-10: 2D working plane geometry.

The following steps describe how to create the 3D geometry by extruding and revolving the 2D geometry:

1 To create the conducting part of the trocar, go to the **Draw** menu and select **Extrude**.

2 In the dialog box that appears, select **C1** in the **Objects to extrude** list and type `10e-3` in the **Distance** edit field.

3 Click **OK**.

4 To create the insulated part of the trocar, return to the Geom2 work plane, then go to the **Draw** menu and select **Extrude**.

5 Select **C1** in the **Objects to extrude** list and type `50e-3` in the **Distance** edit field.

6 Click **OK**.

7 Select the EXT2 geometry object.

8 Click the **Move** button on the Draw toolbar.

9 Enter the following displacement values; when finished, click **OK**.

PROPERTY	VALUE
x	0
y	0
z	10e-3

To create the electrode arms, proceed as follows:

10 Return to the Geom2 work plane.

11 From the **Draw** menu select **Revolve**.

12 From the **Objects to revolve** list select **C2**.

13 Enter the following values in the dialog box; when finished, click **OK**.

PROPERTY	VALUE
$\alpha 1$	0
$\alpha 2$	180
x (point on axis)	-8e-3
y (point on axis)	0
x (second point)	-8e-3
y (second point)	1

14 Revolve the circles C3, C4, and C5 in the same manner using the following values:

- Revolve parameters for the circle C3:

PROPERTY	VALUE
$\alpha 1$	0
$\alpha 2$	-180
x (point on axis)	8e-3
y (point on axis)	0
x (second point)	8e-3
y (second point)	1

- Revolve parameters for the circle C4:

PROPERTY	VALUE
$\alpha 1$	-180
$\alpha 2$	0
x (point on axis)	0
y (point on axis)	-8e-3
x (second point)	1
y (second point)	-8e-3

- Revolve parameters for the circle C5:

PROPERTY	VALUE
$\alpha 1$	180
$\alpha 2$	0
x (point on axis)	0
y (point on axis)	8e-3
x (second point)	1
y (second point)	8e-3

15 To create the blood vessel, return to the Geom2 work plane and select C6.

16 From the **Draw** menu, select **Extrude**.

17 In the **Distance** field, type `120e-3`.

18 Click **OK**.

19 Select the EXT3 geometry object, go to the **Draw** menu, and select **Move** under **Modify**.

20 Enter the following displacement values; when finished, click **OK**.

PROPERTY	VALUE
x	0
y	0
z	-60e-3

21 To create the large cylinder, press the Shift key and then click the **Cylinder** button.

22 Enter the following cylinder properties; when finished, click **OK**.

PROPERTY	VALUE
Radius	0.05
Height	0.12

23 Click the **Zoom Extents** button on the Main toolbar.

This concludes the drawing stage. To get a better view of the geometry you have created, do as follows:

1 To hide the coordinate axes, double-click the **AXIS** button on the status bar at the bottom of the user interface.

2 From the **Options** menu, select **Visualization/Selection Settings**.

3 Clear the **Geometry labels** check box, then click **OK**.

4 Choose **Options>Suppress>Suppress Boundaries**.

5 Select Boundaries 34, 47, 58, 59, 62, and 63, then click **OK**.

6 Click the **Perspective Projection** button on the Camera toolbar.

7 Finally, after rotating the geometry in the drawing area upside down you should have a view similar to that in Figure 4-6 on page 291.

PHYSICS SETTINGS

Subdomain Settings—Bioheat Equation

1 From the **Multiphysics** menu, select **1 Bioheat Equation (htbh)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Select Subdomain 8 and clear the **Active in this domain** check box.

4 Enter the properties for the remaining subdomains as follows:

SETTINGS	SUBDOMAIN 1	SUBDOMAINS 2, 5-7	SUBDOMAIN 3	SUBDOMAIN 4
k	k_l	k_e	k_t	k_c
ρ	rho_l	rho_e	rho_t	rho_c
C	c_l	c_e	c_t	c_c
ρ_b	rho_b			
C_b	c_b			
ω_b	omega_b			
T_b	T_b			
Q_{met}	0			
Q_{ext}	Q_dc			

5 On the **Init** page, select all active subdomains and in the **T(t_0)** edit field type `T0`.

6 To reduce the size of the computation problem, select a lower element order by first clicking the **Element** tab and then selecting **Lagrange - Linear** from the **Predefined elements** list for all active subdomains.

7 Click **OK**.

Boundary Conditions—Bioheat Equation

8 From the **Physics** menu, select **Boundary Settings**.

9 Enter boundary coefficients as follows:

SETTINGS	BOUNDARIES 1-4, 34 47, 58, 59, 62, 63	BOUNDARY 20
Boundary condition	Temperature	Thermal insulation
T_0	T_b	

10 Click **OK**.

Subdomain Settings—Conductive Media DC

1 From the **Multiphysics** menu, select **2 Conductive Media DC (dc)**.

2 From the **Physics** menu, select **Subdomain Settings**.

3 Enter the subdomain properties as in the following table:

SETTINGS	SUBDOMAIN 1	SUBDOMAINS 2, 5–7	SUBDOMAIN 3	SUBDOMAIN 4	SUBDOMAIN 8
σ	sigma_l	sigma_e	sigma_t	sigma_c	sigma_b

4 Reduce the element order also for this application mode: On the **Element** page, select **Lagrange - Linear** from the **Predefined elements** list for all subdomains.

5 Click **OK**.

Boundary Conditions—Conductive Media DC

1 From the **Physics** menu, open the **Boundary Settings** dialog box.

2 Select the **Interior boundaries** check box and enter boundary coefficients as in the table below. You only need to set the boundary condition on the boundaries that use Electric potential. The remaining boundaries already have the correct conditions set by default, that is, Ground for outer boundaries and Continuity for interior boundaries.

SETTINGS	BOUNDARIES 5-16, 21-33, 35-39, 41-43, 45, 46, 48-57	BOUNDARIES 1-4, 20, 34, 47, 60, 61	BOUNDARIES 17-19, 40, 44, 58, 59, 62, 63
Boundary condition	Electric potential	Ground	Continuity
V	V0		

3 Click **OK**.

MESH GENERATION

1 From the **Mesh** menu, select **Free Mesh Parameters**.

2 From the **Predefined mesh sizes** list, select **Fine**.

3 Click the **Custom mesh size** button, then modify the **Element growth rate** from the default value to `1.7`.

4 Click **Remesh**, then click **OK**.

COMPUTING THE SOLUTION

1 Click the **Solver Parameters** button on the Main toolbar.

2 From the **Solver** list, select **Time dependent**.

3 On the **General** page, type `0 600` in the **Times** field.

4 From the **Linear system solver** list, select **Direct (PARDISO)**.

5 On the **Time Stepping** page, select **Time steps from solver** from the **Times to store in output** list.

6 Select the **Manual tuning of step size** check box. Type `0.01` in the **Initial time step** edit field and `50` in the **Maximum time step** edit field.

7 Click **OK**.

8 Click the **Solve** button on the Main toolbar.

POSTPROCESSING AND VISUALIZATION

The default plot is a slice plot of the temperature at time = 600 seconds. In order to generate Figure 4-7, follow these steps:

1 Click the **Plot Parameters** button on the Main toolbar.

2 On the **General** page, select **Interpolated** from the **Solution at time** list.

3 In the **Time** edit field, type 60.

4 Click the **Slice** tab and locate the **Slice data** area. From the **Predefined quantities** list select **Temperature** and from the **Unit** list select **°C**.

5 In the **Slice positioning** area set the number of **x levels**, **y levels**, and **z levels** to 1, 1, and 0, respectively.

6 Click **Apply**.

The following steps describe how to generate the plot in Figure 4-9, which shows the isosurface for the temperature 50 °C after 8 minutes:

1 Return to the **General** page, then clear the **Slice** check box and select the **Isosurface** check box.

2 In the **Time** edit field type 480.

3 Click the **Isosurface** tab. On the **Isosurface Data** page, select **Temperature** from the list of **Predefined quantities** and select **°C** from the **Unit** list.

4 In the **Isosurface levels** area, type 50 in the edit field for **Vector with isolevels**.

5 Click **OK**.

6 To finish the plot, click the **Scene Light** button on the Plot toolbar, then click the **Increase Transparency** button 5–6 times.

   Clicking the **Decrease Transparency** button repeatedly returns the transparency settings to the original state.

To create Figure 4-8, which shows temperature versus time, do the following steps:

1 From the **Postprocessing** menu, select **Domain Plot Parameters**.

2 On the **General** page, click the **Point plot** option button under **Plot type**.

3 On the **Point** page, select Point 43. From the **Unit** list select **°C**, then click **OK**.

The postprocessing image that is shown when you open the tumor ablation model from the model library is created with the following steps:

1 Click the **Plot Parameters** button on the Main toolbar.
2 On the **General** page, clear the **Isosurface** check box and select the **Streamline** and **Slice** check boxes.
3 On the **Streamline** page, select **Heat flux** from the **Predefined quantities** list.
4 On the **Start Points** page, select **Specify start point coordinates**, then specify the following settings in the **x**, **y**, and **z** edit fields:

COORD.	VALUE
x	`linspace(0.02,0.02,30)`
y	`linspace(-0.04,0.04,10) linspace(-0.04,0.04,10) linspace(-0.04,0.04,10)`
z	`linspace(0.05,0.05,10) linspace(0.08,0.08,10) linspace(0.02,0.02,10)`

5 On the **Line Color** page, click the **Use expression** option button, then click the **Color Expression** button. In the dialog box that appears, clear the **Color scale** check box (leave the default settings in the **Streamline color data** area). Click **OK**.
6 Set the **Line type** to **Tube**, then click the **Tube Radius** button. In the dialog box that appears, clear the **Auto** check box for the **Radius scale factor**, then type `0.1` in the edit field. Click **OK**.
7 Click the **Slice** tab. In the **Slice positioning** area set the number of **x levels**, **y levels**, and **z levels** to `1`, `0`, and `0`, respectively.
8 Click **OK**.

INDEX