Introduction to Heat Transfer Module
Introduction to the Heat Transfer Module

© 1998–2013 COMSOL

Protected by U.S. Patents 7,519,518; 7,596,474; and 7,623,991. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/sla) and may be used or copied only under the terms of the license agreement. COMSOL, COMSOL Multiphysics, Capture the Concept, COMSOL Desktop, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/tm.

Version: May 2013

Contact Information

Visit the Contact Us page at www.comsol.com/contact to submit general inquiries, contact Technical Support, or search for an address and phone number. You can also visit the Worldwide Sales Offices page at www.comsol.com/contact/offices for address and contact information.

If you need to contact Support, an online request form is located at the COMSOL Access page at www.comsol.com/support/case.

Other useful links include:

- Support Center: www.comsol.com/support
- Download COMSOL: www.comsol.com/support/download
- Product Updates: www.comsol.com/support/updates
- COMSOL Community: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Center: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM020804
Contents

Introduction ....................................................... 1
Basic Concepts Described in The Heat Transfer Module .... 2
The Applications .................................................. 3
The Heat Transfer Module Interfaces ......................... 8
The Physics List by Space Dimension and Study Type .... 11
The Model Library ............................................... 13
Tutorial Example—Heat Sink ................................. 14
Model Definition .................................................. 14
Results .............................................................. 16
Adding Surface-to-Surface Radiation Effects ................. 28
Introduction

The Heat Transfer Module is used by product designers, developers, and scientists, who use detailed geometrical descriptions to study the influence of heating and cooling in devices or processes. It has modeling tools for the simulation of all mechanisms of heat transfer including conduction, convection, and radiation. Simulations can be run for transient and steady conditions in 1D, 1D axisymmetric, 2D, 2D axisymmetric, and 3D space coordinate systems. The high level detail provided by these simulations allows for the optimization of design and operational conditions in devices and processes influenced by heat transfer.

The Model Library contains tutorials models as well as industrial equipment and device benchmark models for verification and validation.

This introduction fine tunes your COMSOL model building skills for heat transfer simulations. The model tutorial solves a conjugate heat transfer problem from the field of electronic cooling but the principles can be applied to any other field involving heat transfer in solids and fluids.
Basic Concepts Described in The Heat Transfer Module

Heat is one form of energy that, similar to work, is in transit from inside a system or from one system to another. This energy may be stored as kinetic or potential energy in the atoms and molecules in a system.

Conduction is the form of heat transfer that can be described as proportional to the temperature gradients in a system. This is formulated mathematically by Fourier’s law. The Heat Transfer Module describes conduction in systems with constant thermal conductivity and in systems where thermal conductivity is a function of temperature itself or of any other model variable, for example chemical composition.

![Conduction and Convection](image)

Figure 2: Heat transfer in a system containing a solid surrounded by a fluid (conjugate heat transfer). In the fluid, heat transfer can take place through conduction and convection, while conduction is the main heat transfer mechanism in solids. Heat transfer by radiation can occur between surfaces or between surfaces and ambient.

In the case of a moving fluid, the energy transported by the fluid has to be modeled in combination with fluid flow. This is referred to as convection of heat and has to be accounted for in forced and free convection (conduction and advection). This module includes descriptions for heat transfer in fluids and conjugate heat transfer (heat transfer in solids and fluids in the one system) for both laminar and turbulent flows. In the case of turbulent flow the module offers high-Reynolds or, alternatively, low-Reynolds models to accurately describe conjugate heat transfer.

2 | Introduction
Radiation is the third mechanism for heat transfer included in the module. It is modeled using expressions for surface-to-ambient radiation (for example, by defining boundary conditions) and also by using surface-to-surface radiation models, which includes external radiation sources (for example, the sun). The surface-to-surface radiation capabilities are based on the radiosity method. In addition, the module also contains functionality for radiation in participating media. This radiation model accounts for the absorption, emission, and scattering of radiation by the fluid present between radiating surfaces.

The basis of the Heat Transfer Module is the balance of energy in a studied system. The contributions to this energy balance originate from conduction, convection, and radiation but also from latent heat, Joule heating, heat sources, and heat sinks. In the case of moving solids, translational terms may also be included in the heat transfer models, for example for solids in rotating machinery. Effects of solid deformations on thermal properties can also be included. Physical properties and heat sources (or sinks) can be described as arbitrary expressions of the dependent variables in a model (for example, temperature and electric field). The heat transfer equations are defined automatically by the dedicated physics interfaces for heat transfer and fluid flow. The formulations of these equations can be visualized in detail for validation and verification purposes.

Physical properties such as thermal conductivity, heat capacity, density, and emissivity can be obtained from the built-in material library for solids and fluids and from the add-on Material Library in COMSOL. In addition, the module contains relations for the calculation of heat transfer coefficients for different types of convective heat transfer from a surface. For turbulent heat transfer, it also features relations that calculate the thermal conductivity in turbulent flow using the eddy diffusivity from turbulence models (sometimes referred to as turbulent conductivity).

The work flow in the module is straightforward and is described by these steps—define the geometry, select the material to be modeled, select the type of heat transfer, define the boundary and initial conditions, define the finite element mesh, select a solver, and visualize the results. All these steps are accessed from the COMSOL Desktop. The mesh and solver steps are often automatically completed with the default settings, which are already tuned for each type of heat transfer interface.

**The Applications**

Heat generation and transfer are present in most physical processes and phenomena, either as side effects or as desired effects. The module can be effectively used to study a variety of processes, for example to include building ventilation effects, to account for turbulent free convection and heat transfer, to
analyze the impact of electronic microdevice heat generation and cooling, and to study phase change effects.

The Heat Transfer Module’s Model Library contains tutorial and benchmark models from different engineering fields and applications. See “The Model Library” to find out how to access the models.

The Building and Constructions section in the Model Library includes models that are related to energy efficiency and dissipation in buildings. Most of these models use convective heat flux to account for heat exchange between a structure and the surroundings. Simulation provides accurate description of the heat and energy fluxes which facilitates energy management in building and constructions.

![Figure 3: Temperature field in a building wall exposed to cold environment. This plot is from the model Thermal Bridge 3D — Two Floors.](image)

The Heat Exchangers section in the Model Library present several heat exchangers with different sizes, flow arrangements and flow regimes. They benefit from the predefined conjugate heat transfer interface that provides ready to use coupling between solids, shells and laminar or turbulent flows. The simulation results determine the heat exchangers properties like their efficiency, pressure loss or compactness.
The Medical Technology section in the Model Library introduces the concept of bioheating, where the influence of various processes in living tissue are accounted for as contributing to heat flux and as sources and sinks in the heat balance. The types of bioheating applications modeled include hyperthermia cancer therapy, such as microwave heating of tumors, and the interaction between microwave antennas and living tissue, for example, the influence of cellphone use to the temperature of tissue close to the ear. The benefit of using the bioheat equation is that it has been validated for different types of living tissue using empirical data for the different properties, sources, and sinks. The models and simulations defined in this interface provide excellent complements to experimental and clinical trials, which may be used, for example, to develop new methods for dose planning.

The Phase Change section in the Model Library presents applications such as metal melting or food cooking. The common feature with these models is that the temperature field defines the material phase which has a very large impact on the material properties. The highly nonlinear behavior of the material properties as a function of temperature are automatically generated by the heat transfer with phase change feature. The phase change modeling provides information to control material transformation.
The Power Electronics and Electronic Cooling section in the Model Library includes models that often involve heat generation and heat transfer in solids and conjugate heat transfer, where cooling is described in greater detail. The models in these applications are often used to design cooling systems and control the operating conditions of electronic devices and power systems. When the model results are interpreted, it provides the tools needed to understand and optimize the flow and heat transfer mechanisms in these systems.

![Image](image.png)

*Figure 5: Temperature field as a result of conjugate heat transfer in computer Power Supply Unit (PSU). This plot is from the model Electronic Enclosure Cooling.*

The Thermal Contact and Friction section in the Model Library contains examples where the thermal cooling is dependent on a thermal contact or where the heat source is due to friction. The thermal contact properties can be coupled with a structural mechanics that provides the contact pressure at the interface. It is also possible to combine thermal contact and electrical contact in the same model.

The Thermal Processing section in the Model Library has examples including thermal processing of materials such as continuous casting. The common feature with these most of these models is that the temperature field and the temperature variations have a very large impact on the material properties or the physical behavior (thermal expansion, thermophoresis, ...) of the modeled processes and devices. These couplings make these processes difficult to predict and understand.
Modeling and simulation often provide a shortcut to this understanding, which is required to design and operate a system.

![Temperature field plot from the Continuous Casting model. A sharp temperature gradient is found across the mushy layer, where the liquid metal solidifies.](image)

The Thermal Radiation section in the Model Library contains applications where the heat transfer by radiation needs to be considered to describe the heat flux accurately. The common feature between these models is that they contain devices at high temperature which are responsible for high radiative heat transfer. The complexity of such models is due to nonlinearity resulting from radiative heat transfer modeling and the geometrical effects like shielding between two radiating objects, in particular when the geometry is moving or deformed during the simulation.

The Thermal Stress section in the Model Library presents models where the temperature field generates thermal expansion. These models require the Structural Mechanics Module or the MEMS Module for the structural mechanics part. The thermal stress can result from heat exchanges between cold and hot devices or from heat sources like a Joule heating.

The next section describes the available interfaces in this module.
The Heat Transfer Module Interfaces

The figure below shows the Heat Transfer interfaces included in the Heat Transfer Module. These physics interfaces describe different heat transfer mechanisms and also include predefined expressions for sources and sinks. The Heat Transfer interfaces are available in 1D, 2D, 2D axisymmetric, 3D, and for stationary and time-dependent analyses.

HEAT TRANSFER

The Heat Transfer in Solids interface ( ) describes, by default, heat transfer by conduction. It is also able to account for heat flux due to translation in solids, for example, the rotation of a disk or the linear translation of a shaft. It also account for the solid deformation, its volume or surface changes.

The Heat Transfer in Fluids interface ( ) accounts for conduction and convection in gases and liquids as the default heat transfer mechanisms. The coupling to the flow field in the convection term may be entered manually in the user interface or it may be selected from a list that couples heat transfer to an existing fluid flow interface. The Heat Transfer in Fluids interface may be used when the flow field has already been calculated and the heat transfer problem is added afterwards, typically for simulations of forced convection.

The Heat Transfer in Porous Media interface ( ) combines conduction in a porous matrix and in the fluid contained in the pore structure with the convection of heat generated by the flow of the fluid. This physics interface provides a power law or user-defined expression for the effective heat transfer properties and a predefined expression for dispersion in porous media. Dispersion is caused by the tortuous path of the liquid in the porous media, which would not be described if
only the mean convective term was taken into account. This interface may be used for a wide range of porous materials, from porous structures in the pulp and paper industry to the simulation of heat transfer in soil and rock.

The Bioheat Transfer interface is a dedicated interface for heat transfer in living tissue. The bioheat model described in this interface has been verified for different types of living tissue, where also empirical data is available for physical properties, sources, and sinks. Apart from data such as thermal conductivity, heat capacity, and density, tabulated data is also available for blood perfusion rates and metabolic heat sources.

The Heat Transfer in Shells interface contains descriptions for heat transfer where large temperature variations may be present in a 3D structure but where the temperature differences across the thickness of the material of the structure are negligible. Typical examples may be structures like tanks, pipes, heat exchangers, airplane fuselages, and so forth. This physics interface can be combined with other heat transfer interfaces, for example to model the walls of a tank using the Heat Transfer in Thin Shells interface while the Heat Transfer in Fluids feature may be used to model the fluid inside the tank. In many cases using the Highly Conductive Layer boundary condition is the easiest solution.

**Conjugate Heat Transfer**

The Conjugate Heat Transfer interfaces describe heat transfer in solids and fluids and nonisothermal flow in the fluid. The heat transfer process is tightly coupled with the fluid flow problem and the physics interfaces include features for describing heat transfer in free and forced convection, including pressure work and viscous heating. These physics interfaces are available for laminar and turbulent nonisothermal flow. For highly accurate simulations of heat transfer between a solid and a fluid in the turbulent flow regime, low-Reynolds turbulence models resolve the temperature field in the fluid all the way to the solid wall in the Turbulent Flow, Low-Re $k$-$\epsilon$ interface. The standard $k$-$\epsilon$ turbulence model in the Turbulent Flow, $k$-$\epsilon$ interface is computationally inexpensive compared to the other turbulent models but also usually less accurate.

If you also have the CFD Module, three additional turbulent models are available. For less computationally expensive simulations, where wall heat transfer is less important, the $k$-$\omega$ turbulence model gives good accuracy at a comparably low cost. The Spalart-Allmaras interface is a dedicated interface for conjugate heat transfer in aerodynamics, for example for the simulation of wing profiles. The SST (Shear Stress Transport) interface is suitable for many external flows cases or internal flows with sudden expansions.
**Radiation**

The Heat Transfer interface for radiation belong to essentially two different groups of radiation modeling interfaces: the surface-to-surface radiation and the radiation in participating media interfaces. The Heat Transfer with Surface-to-Surface Radiation interface combines heat transfer in fluid or solids including conduction and convection with surface-to-surface radiation. The surface-to-surface radiation accounts for surface properties dependency to the spectral bands. For example, making the difference between ambient radiation (large wavelengths) and the sun radiation (small wavelengths) is necessary to model greenhouse effect. The Heat Transfer with Radiation in Participating Media interface combines heat transfer by conduction and convection in solids and fluids with radiation where absorption or emission of radiation is taken into account by the radiation model. In addition the Surface-to-Surface Radiation interface describes systems where only radiation is computed, typically to estimate radiation between surfaces in space applications where the surface temperature is known. The corresponding Radiation in Participating Media interface computes the radiation, including absorption and emission effects, in a media where the temperature is known.

**Electromagnetic Heating**

The Joule Heating interface is a multiphysics interface that couples electric heating and current conduction in electric conductors with heat transfer for modeling of Joule heating (resistive heating). This multiphysics interface includes the features from the Heat Transfer in Solids interface with the Electric Current interface including predefined couplings for Joule heating.
### The Physics List by Space Dimension and Study Type

The table lists the physics interfaces available with this module in addition to those included with the COMSOL basic license.

<table>
<thead>
<tr>
<th>PHYSICS ICON</th>
<th>PHYSICS</th>
<th>ICON</th>
<th>TAG</th>
<th>SPACE DIMENSION</th>
<th>PRESET STUDY TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image" alt="Fluid Flow" /></td>
<td><strong>Fluid Flow</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Single-Phase Flow" /></td>
<td><strong>Single-Phase Flow</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Single-Phase Flow" /></td>
<td>Single-Phase Flow, Laminar Flow*</td>
<td><img src="image" alt="spf" /></td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Single-Phase Flow" /></td>
<td>Turbulent Flow, k-ε</td>
<td><img src="image" alt="spf" /></td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Single-Phase Flow" /></td>
<td>Turbulent Flow, Low Re k-ε</td>
<td><img src="image" alt="spf" /></td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary with initialization; transient with initialization</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Non-Isothermal Flow" /></td>
<td><strong>Non-Isothermal Flow</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Non-Isothermal Flow" /></td>
<td>Laminar Flow</td>
<td><img src="image" alt="nitf" /></td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Non-Isothermal Flow" /></td>
<td>Turbulent Flow, k-ε</td>
<td><img src="image" alt="nitf" /></td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Non-Isothermal Flow" /></td>
<td>Turbulent Flow, Low Re k-ε</td>
<td><img src="image" alt="nitf" /></td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary with initialization; transient with initialization</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Heat Transfer" /></td>
<td><strong>Heat Transfer</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Heat Transfer" /></td>
<td>Heat Transfer in Solids*</td>
<td><img src="image" alt="ht" /></td>
<td>all dimensions</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Heat Transfer" /></td>
<td>Heat Transfer in Fluids*</td>
<td><img src="image" alt="ht" /></td>
<td>all dimensions</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Heat Transfer" /></td>
<td>Heat Transfer in Porous Media</td>
<td><img src="image" alt="ht" /></td>
<td>all dimensions</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Heat Transfer" /></td>
<td>Bioheat Transfer</td>
<td><img src="image" alt="ht" /></td>
<td>all dimensions</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Heat Transfer" /></td>
<td>Heat Transfer in Thin Shells</td>
<td><img src="image" alt="ht" /></td>
<td>3D</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Conjugate Heat Transfer" /></td>
<td><strong>Conjugate Heat Transfer</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><img src="image" alt="Conjugate Heat Transfer" /></td>
<td>Laminar Flow</td>
<td><img src="image" alt="nitf" /></td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td>PHYSICS</td>
<td>ICON</td>
<td>TAG</td>
<td>SPACE DIMENSION</td>
<td>PRESET STUDY TYPE</td>
<td></td>
</tr>
<tr>
<td>--------------------------------------------------</td>
<td>------</td>
<td>------</td>
<td>--------------------------</td>
<td>--------------------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>Turbulent Flow, k-ε</td>
<td></td>
<td>ntf</td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td>Turbulent Flow, Low Re k-ε</td>
<td></td>
<td>ntf</td>
<td>3D, 2D, 2D axisymmetric</td>
<td>stationary with initialization; transient with initialization</td>
<td></td>
</tr>
<tr>
<td><strong>Radiation</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Heat Transfer with Surface-to-Surface Radiation</td>
<td></td>
<td>ht</td>
<td>all dimensions</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td>Heat Transfer with Radiation in Participating Media</td>
<td></td>
<td>ht</td>
<td>3D, 2D</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td>Surface-to-Surface Radiation</td>
<td></td>
<td>rad</td>
<td>all dimensions</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td>Radiation in Participating Media</td>
<td></td>
<td>rp</td>
<td>3D, 2D</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
<tr>
<td><strong>Electromagnetic Heating</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Joule Heating*</td>
<td></td>
<td>jh</td>
<td>all dimensions</td>
<td>stationary; time dependent</td>
<td></td>
</tr>
</tbody>
</table>

* This is an enhanced interface, which is included with the base COMSOL package but has added functionality for this module.
The Model Library

To open a Heat Transfer Module Model Library model, select View > Model Library from the main menu in COMSOL Multiphysics. In the Model Library window that opens, expand the Heat Transfer Module folder and browse or search the contents. Click Open Model and PDF to open the model in COMSOL Multiphysics and a PDF to read background theory about the model including the step-by-step instructions to build it.

The MPH-files in the COMSOL model libraries can have two formats—Full MPH-files or Compact MPH-files.

- Full MPH-files, including all meshes and solutions. In the Model Library these models appear with the icon. If the MPH-file’s size exceeds 25MB, a tip with the text “Large file” and the file size appears when you position the cursor at the model’s node in the Model Library tree.

- Compact MPH-files with all settings for the model but without built meshes and solution data to save space on the DVD (a few MPH-files have no solutions for other reasons). You can open these models to study the settings and to mesh and re-solve the models. It is also possible to download the full versions—with meshes and solutions—of most of these models through Model Library Update. In the Model Library these models appear with the icon. If you position the cursor at a compact model in the Model Library window, a No solutions stored message appears. If a full MPH-file is available for download, the corresponding node’s context menu includes a Model Library Update item.

A model from the Model Library is used as a tutorial in this guide. See “Tutorial Example—Heat Sink” starting on page 14.
Tutorial Example—Heat Sink

This model is an introduction to simulations of fluid flow and conjugate heat transfer. It demonstrates the following important points. How to:

- Define heat transfer in solids and fluids, including fluid flow.
- Set a total heat source on a domain using automatic volume computation.
- Model the temperature difference between two surfaces in the presence of a thin thermally resistive layer.
- Include the radiative heat transfer between surfaces in a model.

Model Definition

The modeled system consists of an aluminum heat sink for the cooling of an electronic component. See Figure 7

The heat sink is mounted inside a channel of rectangular cross section (see Figure 7). Such a set-up is used to measure the cooling capacity of heat sinks. Air enters the channel at the inlet and exits the channel at the outlet. To improve the thermal contact between the base surface of the heat sink and the top surface of the electronic component, thermal grease is used. All other external faces are thermally insulated. The heat dissipated by the electronic component is equal to 1W and is distributed in all the component volume.

The cooling capacity of the heat sink can be determined by monitoring the temperature in the electronic component.

The model solves a thermal balance for the electronic component, the heat sink, and the air flowing in the rectangular channel. Thermal energy is transported
through conduction in the electronic component and the aluminum heat sink. The temperature field is discontinuous at the interface between the electronic component and the heat sink due to the presence of a thin resistive layer (thermal grease). Thermal energy is transported through conduction and convection in the cooling air. The temperature field is continuous across the internal surfaces between the heat sink and the air in the channel. The temperature is set at the inlet of the channel. A total power of 1 W is dissipated in the electronic component. The transport of thermal energy at the outlet is dominated by convection.

In the first step of the model, heat transfer by radiation between surfaces has been neglected. This assumption is valid as the surfaces have low emissivity (close to 0), which is usually the case for polished metals. When the surface emissivity is large (close to 1), the surface-to-surface radiation should then be considered. This is done in the second step of this tutorial. The model is modified to account for surface-to-surface radiation at the channel and heat sink boundaries. Assuming that the surfaces have been treated with black paint, the surface emissivity is close to 1 in this second case.

The flow field is obtained by solving one momentum balance for each spatial coordinate (x, y, and z) and a mass balance. The inlet velocity is defined by a parabolic velocity profile for fully developed laminar flow. At the outlet, a constant pressure is combined with the assumption that there are no viscous stresses in the direction perpendicular to the outlet. At all solid surfaces, the velocity is set to zero in all three spatial directions.

The thermal conductivity of air, the heat capacity of air, and the air density are all temperature-dependent material properties. You can find all of the settings mentioned in the physics interface for Conjugate Heat Transfer in COMSOL Multiphysics. You also find the material properties, including their temperature dependence, in the Material Browser.
Results
In Figure 8, the hot wake behind the heat sink is a sign of the convective cooling effects. The maximum temperature, reached in the electronic component, is about 377 K.

Figure 8: The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.

In the second step, the temperature and velocity fields are obtained when surface-to-surface radiation is included and the surface emissivities are large. Figure 9 shows that the maximum temperature, about 356 K, is decreased by about 21 K compared to the first case in Figure 8. This confirms that radiative heat
transfer is not negligible when the surface emissivity is close to 1.

Figure 9: Effects of surface-to-surface radiation on temperature and velocity fields. The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.
**Model Wizard**

1. Open COMSOL Multiphysics. In the Model Wizard window the Space Dimension defaults to 3D. Click Next.
2. In the Add physics tree under Heat Transfer>Conjugate Heat Transfer, double-click Laminar Flow (nitf) to add it to the Selected physics list. Click Next.
3. In the Studies tree, under Preset Studies select Stationary.
4. Click Finish.

**Geometry 1**

Follow these steps to import the parameterized model geometry from a separate MPH-file:

1. In the Model Builder window, under Model 1 right-click Geometry 1 and choose Insert Sequence from File.
2. Browse to the model’s Model Library folder and double-click the file heat_sink_geom_sequence.mph. The file containing the sequence is found in COMSOL43b\models\Heat_Transfer_Module\Tutorial_Models,_Forced and Natural Convection\Import 1

**Import 1**

1. In the Model Builder window, under Model 1>Geometry 1 click Import 1.
2. In the Import settings window, locate the Import section.
3. Click the Browse button.
4. Browse to the model’s Model Library folder and double-click the file heat_sink.mphbin.
5. Click the Import button.
6. Click the Build All button.
7. Click the Go to Default 3D View button on the Graphics toolbar.
**Note:** The exact location of the files used in this exercise vary based on the installation. For example, if the installation is on your hard drive, the file path might be similar to `C:\Program Files\COMSOL43b\models\Heat_Transfer_Module\Tutorial_Models`

---

**Global Definitions**

**Parameters**
1. In the Model Builder window, under Global Definitions click Parameters.
2. In the Parameters settings window, locate the Parameters section.
3. In the table, enter the following settings:

<table>
<thead>
<tr>
<th>NAME</th>
<th>EXPRESSION</th>
<th>DESCRIPTION</th>
</tr>
</thead>
<tbody>
<tr>
<td>U0</td>
<td>5 [cm/s]</td>
<td>Mean inlet velocity</td>
</tr>
<tr>
<td>T0</td>
<td>20 [degC]</td>
<td>Inlet temperature</td>
</tr>
<tr>
<td>P_tot</td>
<td>1 [W]</td>
<td>Total power dissipated by the electronic package</td>
</tr>
</tbody>
</table>

Together with the parameters contained in the geometry sequence file, the parameter list should look as follows:

<table>
<thead>
<tr>
<th>Name</th>
<th>Expression</th>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>L_channel</td>
<td>7 [cm]</td>
<td>0.070000 m</td>
<td>Channel length</td>
</tr>
<tr>
<td>W_channel</td>
<td>8 [cm]</td>
<td>0.080000 m</td>
<td>Channel width</td>
</tr>
<tr>
<td>H_channel</td>
<td>1.5 [cm]</td>
<td>0.015000 m</td>
<td>Channel height</td>
</tr>
<tr>
<td>L_chip</td>
<td>1.5 [cm]</td>
<td>0.015000 m</td>
<td>Chip size</td>
</tr>
<tr>
<td>H_chip</td>
<td>2 [mm]</td>
<td>0.00020000 m</td>
<td>Chip height</td>
</tr>
<tr>
<td>U0</td>
<td>5 [cm/s]</td>
<td>0.050000 m/s</td>
<td>Mean inlet velocity</td>
</tr>
<tr>
<td>T0</td>
<td>20 [degC]</td>
<td>293.15 K</td>
<td>Inlet temperature</td>
</tr>
<tr>
<td>P_tot</td>
<td>1 [W]</td>
<td>1.0000 W</td>
<td>Total power dissipated by the electronic package</td>
</tr>
</tbody>
</table>
**Materials**

To facilitate face selection in the next steps, use the wireframe rendering option.

*Air and Aluminum 3003-H18*

1. Click the WireFrame Rendering button on the Graphics toolbar.
2. From the main menu, select View>Material Browser.
3. In the Material Browser window in the Materials tree under Built-In, right-click Air and choose Add Material to Model.

By default, the first material added applies to all domains. Typically, you can leave this setting and add other materials that override the default material where applicable. In this example, specify aluminum for Domain 2.

4. Go to the Material Browser window.
5. In the Materials tree under Built-In, right-click Aluminum 3003-H18 and choose Add Material to Model.

7. In the Graphics window, select Domain 2 only and add it to the Selection list.

*Silica Glass*

1. Go to the Material Browser. In the Materials tree under Built-In, right-click Silica glass and choose Add Material to Model.
2. In the Model Builder, click Silica glass.
3. Select Domain 3 only.
**Thermal Grease**

1. In the Model Builder, right-click Materials and choose Material.
2. In the Material settings window, locate the Geometric Entity Selection section.
3. From the Geometric entity level list, choose Boundary. Select Boundary 34 only.
4. Right-click Material 4 and choose Rename (or press F2). Enter Thermal Grease in the New name field. Click OK.
5. In the Material settings window, click to expand the Material Properties section.
6. In the Material properties tree, select Basic Properties>Thermal Conductivity.
7. Click the Add to Material button under the table.

8. Under the Material Contents section, in the table, enter the following settings:

<table>
<thead>
<tr>
<th>Property</th>
<th>Name</th>
<th>Value</th>
<th>Unit</th>
<th>Property group</th>
</tr>
</thead>
<tbody>
<tr>
<td>Thermal conductivity</td>
<td>k</td>
<td>2[W/m/K]</td>
<td>W/(m/K)</td>
<td>Basic</td>
</tr>
</tbody>
</table>

The final node sequence under Materials should match this figure.
Conjugate Heat Transfer (nitf)

Now define the physical properties of the model. Start by adding a Fluid feature to define the fluid domain.

**Fluid 1**
1. In the Model Builder, under Conjugate Heat Transfer (nitf), click Fluid 1.
2. In the Fluid settings window select Domain 1 only.

Next use the power parameter to define the total heat source equal in the electronic package.

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

2. In the Heat Source settings window, under Domain Selection, select Domain 3 only.
3. Under the Heat Source section, click the Total power button. In the $P_{tot}$ field, enter $P_{tot}$.

For the default Wall node, No slip is the default boundary condition for the fluid. Define the inlet and outlet conditions as described below.
**Inlet**

1. In the Model Builder, right-click Conjugate Heat Transfer (nitf) and choose the boundary condition Laminar Flow> Inlet.
2. In the Inlet settings window, select Boundary 121 only.
3. Under Boundary Condition from the Boundary condition list, select Laminar inflow.
4. Under Laminar Inflow in the $U_{av}$ field, enter $U_0$.

**Outlet**

1. In the Model Builder, right-click Conjugate Heat Transfer (nitf) and choose the boundary condition Laminar Flow>Outlet.
2. Click the Zoom Extents button on the Graphics toolbar.
3. In the Outlet settings window select Boundary 1 only.

The node sequence in the Model Builder should match this figure so far. The ‘D’ in the upper left corner of a node means it is a default node.

Thermal insulation is the default boundary condition for the temperature. Define the inlet temperature and the outlet condition.
**Temperature 1**

1. In the Model Builder, right-click Conjugate Heat Transfer (nitf) and from the boundary level choose Heat Transfer>Temperature.

2. In the Temperature settings window select Boundary 121 only.

3. Under the Temperature section in the $T_0$ field, enter $T_0$.

**Outflow 1**

1. In the Model Builder, right-click Conjugate Heat Transfer (nitf) and choose the boundary condition Heat Transfer>Outflow.

2. In the Outflow settings window select Boundary 1 only.

**Thin Thermally Resistive Layer 1**

1. In the Model Builder, right-click Conjugate Heat Transfer (nitf) and choose the boundary condition Heat Transfer>Thin Thermally Resistive Layer.

2. In the Thin Thermally Resistive Layer settings window select Boundary 34 only.
3 Under Thin Thermally Resistive Layer in the $d_n$ field, enter $50 \text{[um]}$.

The node sequence in the Model Builder under Conjugate Heat Transfer should match the figure.

**Mesh 1**

*Free Tetrahedral 1 and Size*

1 In the Model Builder under Model 1 (mod1), click Mesh 1 $\mathbb{C}$.

2 Go to the Mesh settings window. Under Mesh Settings from the Element Size list, select Extra coarse.

3 Click the Build All button $\mathbb{F}$.

To get a better view of the mesh, suppress some of the boundaries.
4 In the Graphics window, select Boundaries 1, 2, and 4 only. Click the Hide Selected button on the toolbar. The mesh is generated and displayed as shown in the figure below.

To get more accuracy in the numerical results, this mesh can be refined by choosing another predefined element size. However, doing so requires more computational time and memory.

**Study 1**

1 In the Model Builder, right-click Study 1 and choose Compute. Two default plots are generated automatically. The first one shows the velocity magnitude on five parallel slices. The second one shows the temperature on the wall boundaries. Add an arrow plot to visualize the velocity field.
Results

Temperature \((\text{nitf})\)

1. Under Results right-click Temperature \((\text{nitf})\) and choose Arrow Volume. You notice that the velocity field is represented by default.

2. Go to the Arrow Volume settings window. Under Data from the Data set list, select Solution 1.

3. Under Arrow Positioning:
   - In the x grid points Points field, enter 20.
   - In the y grid points Points field, enter 10.
   - In the z grid points subsection from the Entry method list, select Coordinates.
   - In the Coordinates field, enter \(5 \times 10^{-3}\) or \(5\text{[mm]}\).

4. In the Model Builder, right-click Arrow Volume 1 and choose Color Expression.

5. Go to the Color Expression settings window. In the upper-right corner of the Expression section, click Replace Expression.

6. From the menu, choose Conjugate Heat Transfer (Laminar Flow)>Velocity magnitude \((\text{nitf.U})\) (or enter \texttt{nitf.U} in the Expression field).

7. Click the Plot button.

The plot in Figure 8 on page 16 is displayed in the Graphics window.
Adding Surface-to-Surface Radiation Effects

Now we can modify the model to include surface-to-surface radiation effects. First we need to enable surface to surface property in the physics interface. Then, we will study the effects of surface to surface radiation between the heat sink and the channel walls.

Conjugate Heat Transfer

You can continue using the model built so far, or you can open the model from the Model Library.

1. To open the model in the Model Library, select View>Model Library. Under the Heat Transfer Module>Tutorial_Models_Forced and Natural Convection folder, click heat_sink and click Open.

2. In the Model Builder, under Model 1, click Conjugate Heat Transfer.

3. In the Conjugate Heat Transfer settings window, under the Physical Model section, click to select the Surface-to-surface radiation check box.

Now it is possible to add the surface to surface boundary condition to the model.

Surface-to-Surface Radiation

1. Right-click Conjugate Heat Transfer and choose the boundary condition Surface-to-Surface Radiation>Surface-to-Surface Radiation.

A Surface-to-Surface Radiation node with a second default Initial Values node is added to the Model Builder.

2. In the Model Builder, click the Surface-to-Surface Radiation 1 node.
3 Select Boundaries 6, 7, 9–30, 36–109, and 111–120 only.

Note: There are many ways to select geometric entities. When you know the domain to add, such as in this example, you can click the Paste Selection button and enter the information in the Selection field. For more information about selecting geometric entities in the Graphics window, see the COMSOL Multiphysics Reference Manual.

4 In the Surface-to-Surface Radiation settings window, under Surface Emissivity, from the $\varepsilon$ list, choose User defined. Enter 0.85 in the Surface emissivity edit field.

By default the radiation direction is controlled by the opacity of the domains. The solid parts are automatically defined as opaque while the fluid parts are transparent. You can change these settings by modifying or adding an Opaque subnode under Heat Transfer in Solids and Fluid nodes.

Surface-to-Surface Radiation 2

1 Add another Surface-to-Surface Radiation node. In the Model Builder, right-click Conjugate Heat Transfer and choose the boundary condition Surface-to-Surface Radiation>Surface-to-Surface Radiation.

2 Select Boundaries 2–5 only.

3 In the Surface-to-Surface Radiation settings window, under Surface Emissivity from the $\varepsilon$ list, choose User defined. Enter 0.9 in the Surface emissivity edit field.

4 Under Radiation Settings, from the Radiation direction list, choose Negative normal direction. This setting defines the channel interior as transparent and the exterior as opaque.
Model Wizard

In order to keep the previous solution and to be able to compare it with this version of the model, add a second stationary study.

1. In the Model Builder, right-click the Root node and choose Add Study.


3. Click Finish.

Study 2

Step 1: Stationary

1. In the Model Builder, right-click Study 2 and choose Compute.

Results

The same default plots are generated as in “Results” on page 27. In this step, you can edit the temperature plot to compare the results.

Temperature (nitf)

1. Under Results right-click Temperature (nitf) and choose Arrow Volume. You notice that the velocity field is represented by default.

2. Under Data from the Data set list, choose Solution 2.
3 Under Arrow Positioning:
- In the x grid points Points field, enter 20.
- In the y grid points Points field, enter 10.
- In the z grid points subsection from the Entry method list, select Coordinates.
- In the Coordinates field, enter $5 \times 10^{-3}$ or 5 [mm].

4 Under Results, right-click Arrow Volume 1 and choose Color Expression.

5 In the Color Expression settings window click the Replace Expression button.

6 From the menu, choose Conjugate Heat Transfer (Laminar Flow) > Velocity magnitude (nitf_U).

7 Click the Plot button. The plot that displays should look the same as below and Figure 9 on page 17.