INTRODUCTION TO

Heat Transfer Module

5.2a

COMSOL

Introduction to the Heat Transfer Module

Version: COMSOL 5.2a

Contact Information

Visit the Contact COMSOL 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
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/support/updates
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/community
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/video
- Support Knowledge Base: www.comsol.com/support/knowledgebase

Part number: CM020804

Contents

Introduction

The Heat Transfer Module is used by product designers, developers, and scientists, who use detailed geometric models to study the influence of heating and cooling in devices and processes. It contains 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 coordinate systems.

The high level of detail provided by these simulations allows for the optimization of design and operational conditions in devices and processes influenced by heat transfer.

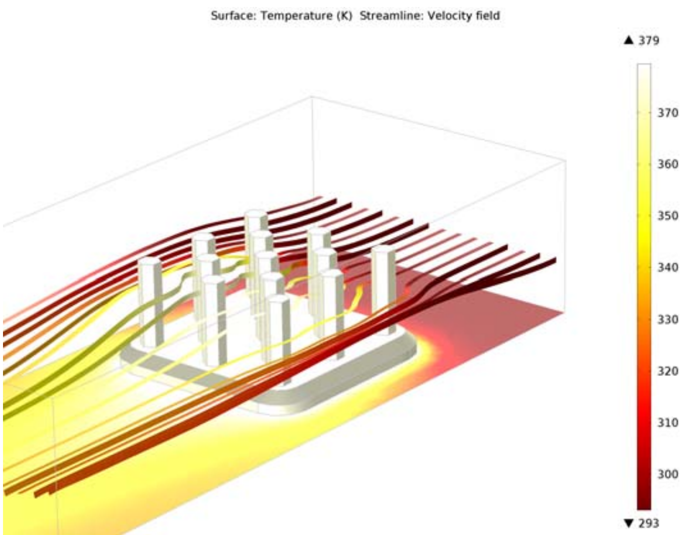


Figure 1: Temperature and flow field in an aluminum heat sink and in cooling air that is pumped over the heat sink. The temperature and flow field are solved using detailed geometry and a description of the physics.

The Application Libraries window contain tutorials, as well as industrial equipment and device benchmark applications for verification and validation.

This introduction fine-tunes your COMSOL modeling skills for heat transfer simulations. The model tutorial solves a conjugate heat transfer problem from the field of electronic cooling, but the principles are applicable to any field involving heat transfer in solids and fluids.

Basic Concepts Described in The Heat Transfer Module

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

Conduction is a 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 where thermal conductivity is constant, or is a function of temperature or any other model variable, for example chemical composition.

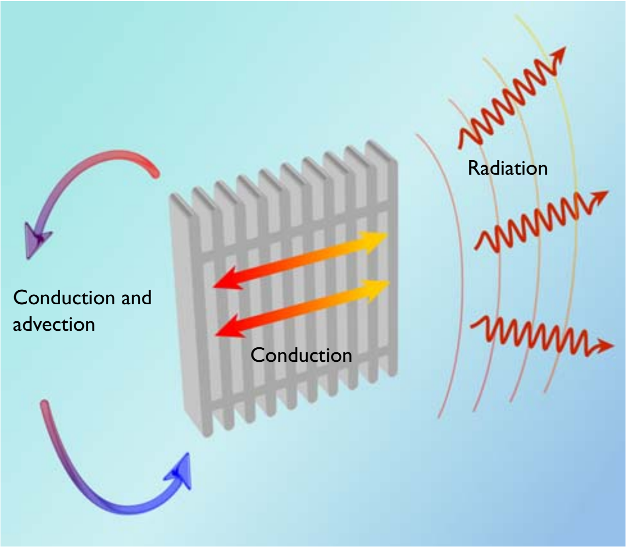


Figure 2: Heat transfer in a system containing a solid surrounded by a fluid (conjugate heat transfer). In the fluid, heat transfer can take place through conduction and advection, while in a solid, conduction is the main heat transfer mechanism. Heat transfer by radiation can occur between surfaces,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

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 must 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 same system), for both laminar and turbulent flows. In the case of turbulent flow, the module offers two algebraic turbulence models, the Algebraic yPlus and L-VEL models, as well as the standard k-ε model for high-Reynolds and a low Reynolds number k-ε model to accurately describe conjugate heat transfer. The gravity feature defines buoyancy forces induced by density differences, in particular due to temperature dependency of the density.

Radiation is the third mechanism for heat transfer included in the module. The associated features handle surface-to-ambient radiation, surface-to-surface radiation, and also external radiation sources (for example, the sun). The surface-to-surface radiation capabilities are based on the radiosity method. It is also possible to combine the Heat Transfer Module with the Particule Tracing Module to model mixed diffuse-specular reflection. In addition, the Heat Transfer 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 module offers three models for participating media simulations: the discrete ordinate method (DOM), the P1 method and the Rosseland approximation.

The basis of the Heat Transfer Module is the study of the balance of energy in a 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. The effects of solid deformations on thermal properties can also be modeled. Physical properties and heat sources (or sinks) can be described as arbitrary expressions containing 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 used to 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 defined by the following 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 included with the default settings, which are tailored to 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 Heat Transfer Module can be effectively used to study a variety of processes (for example, building ventilation effects); to account for turbulent free convection and heat transfer; to analyze the impact of heat generation and cooling in electronic microdevices; and to study phase change effects.

The Heat Transfer Module's application library contains tutorial and benchmark models from different engineering applications. See Tutorial Example — Heat Sink to find out how to access the applications.

The Applications section in the application library contains ready made applications than can be run using a dedicated interfaces. These applications are design to solve a particular class problems and come a simplified GUI that make it easy to use, even for non-COMSOL Multiphysics users.

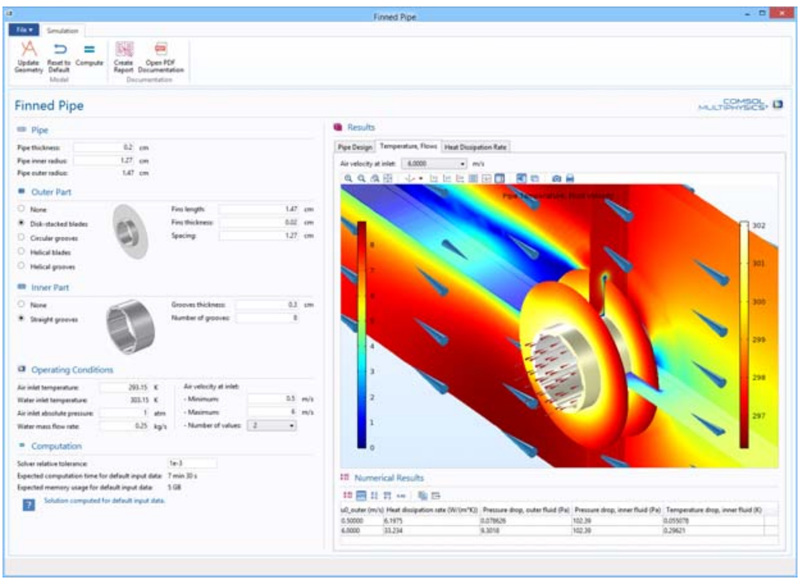


Figure 3: Snapshot of an application dedicated to finned pipe characterization. The applications computes finned pipe properties such as the pressure drop and the temperature drop in the pipe. The finned pipe design and the operating conditions are customizable via a dedicated GUI.

The Building and Constructions section in the application library includes files that are related to energy efficiency and dissipation in buildings. Most of these use convective heat flux to account for heat exchange between a structure and its surroundings. Simulation provides accurate depictions of the heat and energy fluxes that inform energy management in buildings and construction.

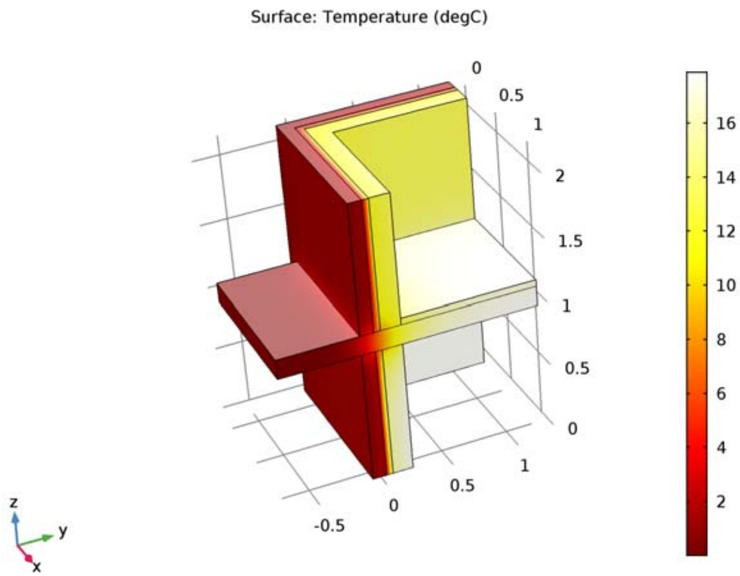


Figure 4: Temperature field in a building wall exposed to a cold environment. This plot is from the model Thermal Bridge 3D — Two Floors.

The Heat Exchangers section in the application library presents several heat exchangers of different sizes, flow arrangements, and flow regimes. They benefit from the predefined conjugate heat transfer interface that provides ready-to-use features for couplings between solids, shells, and laminar or turbulent flows. The simulation results show properties of the heat exchangers, such as their efficiency, pressure loss, or compactness.

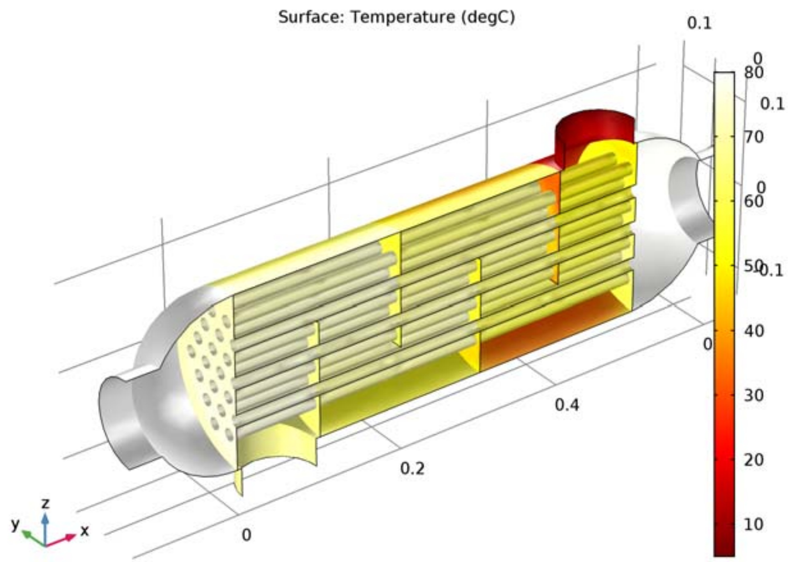


Figure 5: Wall temperature in a shell-and-tube heat exchanger resulting from heat exchange between a cold and a hot fluid separated by a thin wall. This plot is from the Shell-and-Tube Heat Exchanger model.

The Medical Technology section in the application library introduces the concept of bioheating. Here, the influences of various processes in living tissue are accounted for as contributions to heat flux, and as sources and sinks in the heat balance relations. Bioheating applications that can be modeled include the microwave heating of tumors (such as hyperthermia cancer therapy), and the interaction between microwave antennas and living tissue (such as the influenceo f a diagnostic probe or of cellphone use on 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. In addition, damage-integral features are provided to model tissue necrosis due to hyperthermia or hypothermia. The models and simulations available in this physics interface provide excellent complements to experimental and clinical trials; the results may be used for many purposes, for example, to develop new methods for dose planning.

The Phase Change section in the application library presents applications such as metal melting, evaporation and food cooking. A common characteristic of these models is that the temperature field defines the material phase, which has a significant impact on the material properties. Equations representing 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 model provides information to control material transformation.

The Power Electronics and Electronic Cooling section in the application library includes examples that often involve heat generation, heat transfer in solids, and conjugate heat transfer (where cooling is described in greater detail). The examples in these applications are often used to design cooling systems and to control the operating conditions of electronic devices and power systems. The module provides the tools needed to understand and optimize flow and heat transfer mechanisms in these systems when the model results are interpreted.

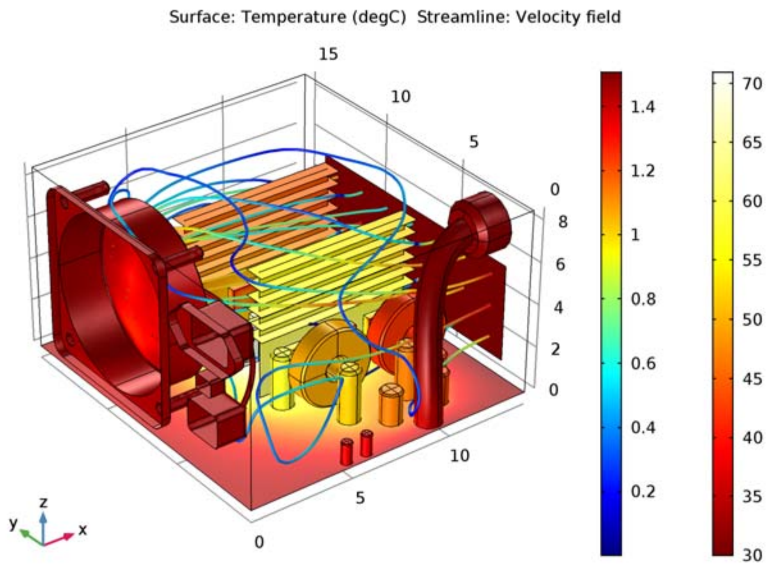


Figure 6: Temperature field and flow streamlines as a result of conjugate heat transfer in computer power supply unit (PSU). This plot is from the model Electronic Enclosure Cooling.

The Thermal Contact and Friction section in the application library contains examples where 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 structural mechanics that provide 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 application library has examples that include thermal processes, such as continuous casting. A common characteristic of most of these examples is that the temperature field and the temperature variations have a significant impact on the material properties or the physical behavior (thermal expansion, thermophoresis, and so forth) of the modeled process or device. Since these couplings make the processes very complicated, modeling and simulation often provide a useful shortcut to a complex understanding.

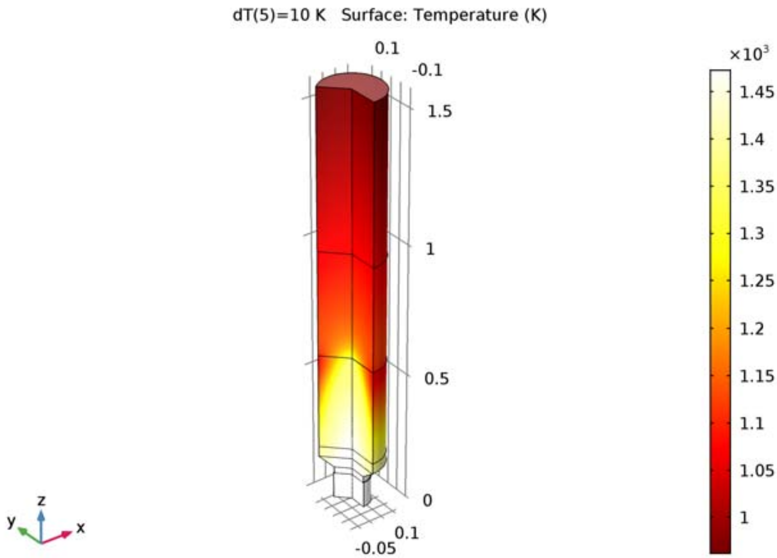


Figure 7: Temperature field plot from the Continuous Casting model. A sharp temperature gradient is found across the mushy layer, where the liquid metal solidifies.

The Thermal Radiation section in the application library contains applications where heat transfer by radiation must be considered in order to describe the heat flux accurately. A common feature of these examples is that they contain devices at high temperatures, which are responsible for high radiative heat transfer. The nonlinearity resulting from the radiative heat transfer, as well as geometric effects such as shielding between two radiating objects, makes such applications quite complex. They become even more so when geometry is moving or deformed during the simulation.

The Thermal Stress section in the application library presents examples where the temperature field causes thermal expansion. Thermal stress can result from heat exchanges between cold and hot devices or from processes like Joule heating. These require the Structural Mechanics Module or the MEMS Module for the portions where structural mechanics are simulated.

The Tutorials sections in the application library contains examples that demonstrate the implementation of a particular phenomenon or the use of some features. Reproducing these applications is an effective method to discover the capabilities of the Heat Transfer module and to get more experience using it.

The Verification Examples sections in the application library provides examples reproducing a case with a know solution and compare it COMSOL Multiphysics results.

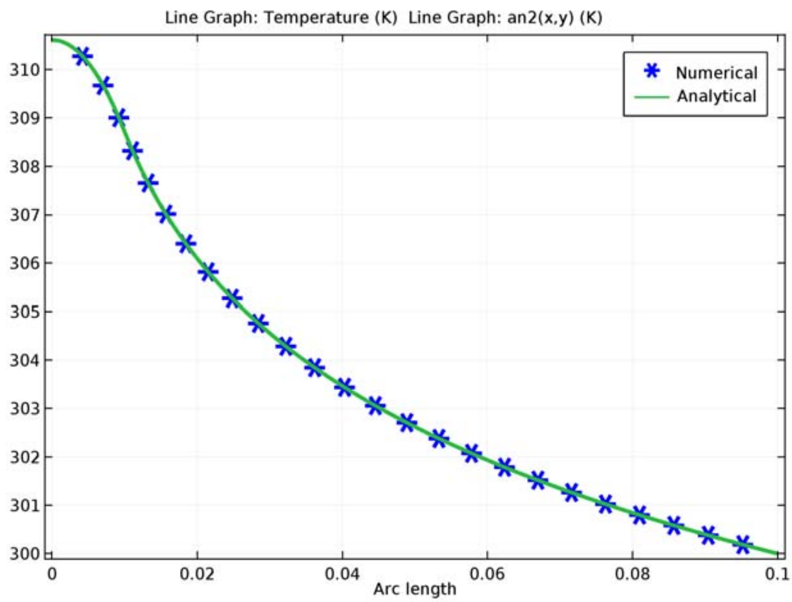


Figure 8: Comparison of the temperature obtained by COMSOL Multiphysics (green line) with an analytical solution.

The next section describes the available physics interfaces in this module.

The Heat Transfer Module Physics 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 for 1D, 2D, 2D axisymmetric, and 3D coordinate systems, and with stationary and time-dependent analyses.

- Chemical Species Transport
  - Transport of Diluted Species (tds)
  - Moisture Transport (mt)
- Fluid Flow
  - Single-Phase Flow
    - Laminar Flow (spf)
    - Turbulent Flow
      - Turbulent Flow, Algebraic yPlus (spf)
      - Turbulent Flow, L-VEL (spf)
      - Turbulent Flow, k-ε (spf)
      - Turbulent Flow, Low Re k-ε (spf)
  - Non-Isothermal Flow
    - Laminar Flow
    - Turbulent Flow
      - Turbulent Flow, Algebraic yPlus
      - Turbulent Flow, L-VEL
      - Turbulent Flow, k-ε
      - Turbulent Flow, Low Re k-ε
- Heat Transfer
  - Heat Transfer in Solids (ht)
  - Heat Transfer in Fluids (ht)
  - Local Thermal Non-Equilibrium
  - Heat Transfer in Porous Media (ht)
  - Bioheat Transfer (ht)
  - Heat and Moisture Transport
  - Thin Structures
    - Heat Transfer in Thin Shells (htsh)
    - Heat Transfer in Thin Films (htsh)
    - Heat Transfer in Fractures (htsh)
  - Conjugate Heat Transfer
    - Laminar Flow
    - Turbulent Flow
      - Turbulent Flow, Algebraic yPlus
      - Turbulent Flow, L-VEL
      - Turbulent Flow, k-ε
      - Turbulent Flow, Low Re k-ε
  - Radiation
    - Heat Transfer with Surface-to-Surface Radiation (ht)
    - Heat Transfer with Radiation in Participating Media (ht)
    - Surface-to-Surface Radiation (rad)
    - Radiation in Participating Media (rpm)
  - Electromagnetic Heating
    - Joule Heating
  - Thermoelectric Effect

Heat Transfer

The Heat Transfer in Solids interface ( ) describes, by default, heat transfer by conduction. It can also account for heat flux due to translation in solids (for example, the rotation of a disk or the linear translation of a shaft), as well as for solid deformation, including 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 physics 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 Local Thermal Non-Equilibrium (LTNE) multiphysics interface ( ) is macro-scale model designed to simulate heat transfer in porous media where the temperatures into the porous matrix and the fluid are not in equilibrium. It differs from simpler macro-scale models for heat transfer in porous media where temperature difference of the solid and fluid are neglected. The absence of thermal equilibrium can result from fast transient changes but can also be observed in stationary cases. Typical applications are rapid heating or cooling of a porous media using a hot fluid or internal heat generation in one of the phases (due to inductive or microwave heating, exothermic reactions, etc.). This is observed in nuclear devices, electronics system or fuel cells for example.

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 uses the provided power law or a 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. (This would be absent if the mean convective term was accounted for.) This physics 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. In addition to data such as thermal conductivity, heat capacity, and density, tabulated data is available for blood perfusion rates and metabolic heat sources. Tissue damage integral models based on a temperature threshold or an energy absorption model can also be included.

The Heat and Moisture Transport interface ( ) combines the Heat Transfer in Building interface with the Moisture Transport interface. It can be used to model different moisture variations phenomena in building components, such as drying of initial construction moisture, condensation due to migration of moisture from outside to inside, or moisture accumulation by interstitial condensation due to diffusion.

The Thermoelectric interface ( ) combine the Electric Currents and the Heat Transfer in Solids interfaces with capabilities for modeling thermoelectric effect (Peltier-Seebeck-Thomson effects) as well as Joule heating (resistive heating). This multiphysics coupling accounts for Peltier heat source or sink and resistive losses in the Heat Transfer interfaces, as well as for the current induce by the Seebeck effect and for the temperature dependency of material properties in the Electric Currents interface.

This physics interface is automatically paired with the AC/DC module capabilities for advanced modeling of electric effects.

Thin Structures

The Thin Structures interfaces ( ) provides efficient models defined at the boundaries level but that represent thin three-dimensional domains. Three different interface come with different default features.

The Heat Transfer in Thin Shells interface ( ) contains descriptions for heat transfer in shell structures where large temperature variations may be present. Thin conductive shells correspond to the simplest model where the shell is represented as homogeneous and the temperature differences across the thickness of the structure material is neglected. Thin layered shells can represent multi-layered structure with heterogeneous material properties and compute the temperature variation across the shell sides. Typical examples of these structures are tanks, pipes, heat exchangers, airplane fuselages, and so forth. This physics interface can be combined with other Heat Transfer interfaces. For example, the Heat Transfer in Thin Shells interface may be used to model the walls of a tank while the Heat Transfer in Fluids interface may be used to model the fluid inside the tank. In many cases, using the Thin Layer boundary condition, found in the Heat Transfer interfaces, produces the easiest solution.

The Heat Transfer in Thin Films interface ( ) implements a model to describe the temperature field in films. The simplest model for thermally thin films assumes that the temperature changes through the film thickness can be neglected. This computationally effective model is sufficient in many cases. The general model computes the temperature variation across the film. Typical applications are when the sides of the film are exposed to different temperature or when heat is dissipated in the film.

The Heat Transfer in Fractures interface ( ) describes heat transfer in a thin porous media assuming the temperature changes through the film thickness can be neglected.

Conjugate Heat Transfer

The Conjugate Heat Transfer interfaces ( ) combine all features from the Heat Transfer and Single-Phase Flow interfaces to describe heat transfer in solids and fluids, and nonisothermal flow in fluids. The heat transfer process is tightly coupled with the fluid flow problem via a predefined multiphysics coupling. These interfaces are available for laminar, turbulent nonisothermal flow and flow in porous media (Brinkman equation). 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. This model is available in the Turbulent Flow, Low-Re k-ε interface ( ). The standard k-ε turbulence model in the Turbulent Flow, k-ε interface ( ) is computationally inexpensive compared to other transport two equation turbulence models, but usually less accurate. The Algebraic yPlus and L-VEL interfaces are adapted for internal flows.

With the use of the CFD Module, three additional turbulence models are available. The k-ω model is an alternative to the standard k-ε model and often gives more accurate results, especially in recirculation regions and close to solid walls. However, the k-ω model is also less robust than the standard k-ε model. The Spalart-Allmaras interface is a dedicated physics interface for conjugate heat transfer in aerodynamics, for example in the simulation of wing profiles. The SST (Shear Stress Transport) interface is suitable for many external flow cases and internal flows with sudden expansions.

Radiation

The Heat Transfer interfaces for radiation essentially belong to two different groups of radiation modeling: surface-to-surface radiation and the radiation in participating media. The Heat Transfer with Surface-to-Surface Radiation interface ( ) combines heat transfer in fluids or solids, including conduction and convection with surface-to-surface radiation. The surface-to-surface radiation model also accounts for the dependency of surface properties on the spectral bands. For example, to model the greenhouse effect it is necessary to solve separately for ambient radiation (large wavelengths) and the sun's radiation (small wavelengths). The Heat Transfer with Radiation in Participating Media interface ( ) combines conduction and convection in solids and fluids with radiation where absorption or emission of radiation is accounted for by the radiation model. 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 ( ) can combine the Electric Currents and the Heat Transfer in Solids interfaces with capabilities for modeling Joule heating (resistive heating). This multiphysics coupling accounts for electromagnetic losses in the Heat Transfer interfaces, as well as for the temperature dependency of material properties in the Electric Currents interface.

Physics Interface Guide 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.

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Chemical Species Transport				
Moisture Transport		mt	all dimensions	stationary; time dependent
Fluid Flow				
Single-Phase Flow				
Laminar Flow[1]		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, Algebraic yPlus		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Non-Isothermal Flow				
Laminar Flow(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, Algebraic yPlus(2)		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL(2)		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε(2)		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Heat Transfer				
Heat Transfer in Solids[1]		ht	all dimensions	stationary; time dependent
Heat Transfer in Fluids(1)		ht	all dimensions	stationary; time dependent
Local Thermal Non-Equilibrium[2]		—	all dimensions	stationary; time dependent
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent
Heat and Moisture Transport		—	all dimensions	stationary; time dependent
Bioheat Transfer		ht	all dimensions	stationary; time dependent
Thermoelectric Effect[2]		—	all dimensions	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Thin Structures				
Heat Transfer in Thin Shells		htsh	3D, 2D, 2D axisymmetric	stationary; time dependent
Heat Transfer in Thin Films		htsh	3D, 2D, 2D axisymmetric	stationary; time dependent
Heat Transfer in Fractures		htsh	3D, 2D, 2D axisymmetric	stationary; time dependent
Conjugate Heat Transfer				
Laminar Flow(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow				
Turbulent Flow, Algebraic yPlus		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε(2)		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε(2)		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Radiation				
Heat Transfer with Surface-to-Surface Radiation		ht	all dimensions	stationary; time dependent
Heat Transfer with Radiation in Participating Media		ht	3D, 2D	stationary; time dependent
Surface-to-Surface Radiation		rad	all dimensions	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Radiation in Participating Media		rpm	3D, 2D	stationary; time dependent
Electromagnetic Heating				
Joule Heating[1,2]		—	all dimensions	stationary; time dependent;
(1) This physics interface is included with the core COMSOL package but has added functionality for this module.				
(2) This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.				

Tutorial Example — Heat Sink

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

- Defining heat transfer in solids and fluids, including fluid flow
- Setting a total heat source on a domain using automatic volume computation
- Modeling the temperature difference between two surfaces when a thin thermally resistive layer is present
- Including the radiative heat transfer between surfaces in a model

Model Definition

The modeled system describes an aluminum heat sink used for the cooling of an electronic component as shown in Figure 9

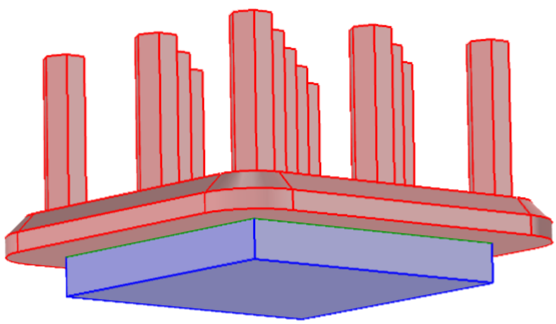


Figure 9: Heat sink and electronic component geometry.

The heat sink represented in Figure 9 (pink in the figure) is mounted inside a channel with a rectangular cross section. Such a setup 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 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 1 W and is distributed through 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 transfered by conduction in the electronic component and the aluminum heat sink. Thermal energy is transported by conduction and advection in the cooling air. 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 (the thermal grease). The temperature field is continuous across all other internal surfaces. The temperature is set at the inlet of the channel. 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. In a case where the surface emissivity is large (close to 1), the surface-to-surface radiation should be considered. This is done in the second step of this tutorial, where 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 relation for each space coordinate (x, y, and z) and a mass balance equation. The inlet velocity is defined by a parabolic velocity profile for fully developed laminar flow. At the outlet, the normal stress is equal to the outlet pressure and the tangential stress is canceled. 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 10, 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 374 K.

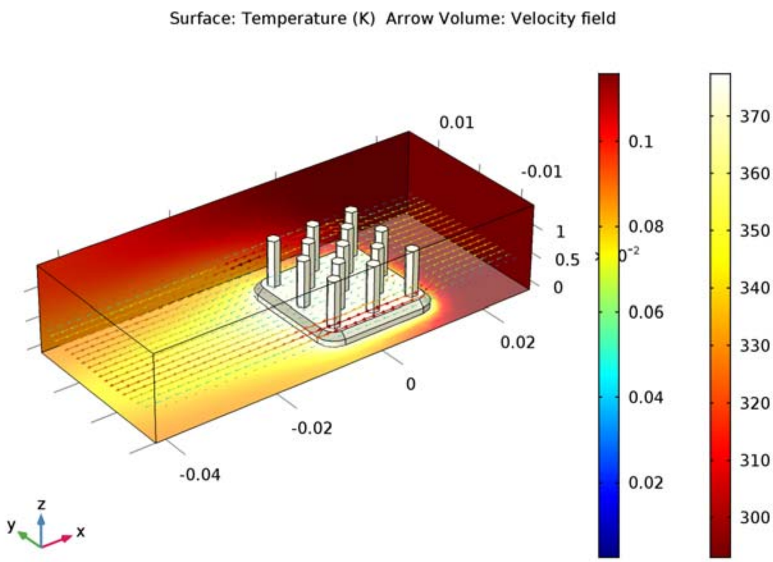


Figure 10: The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.

The energy balance can be checked to verify the consistency of the model. Because the study is stationary, there is no accumulation term. So, the energy balance is performed by comparing the energy flux through all exterior boundaries of the system with the sum of all heat sources. Numerical evaluation of corresponding quantities correspond, respectively, to 0.99998 W and 1 W. Since these two values are extremely close, the energy balance is validated.

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 11 shows that the maximum temperature, about 354 K, is decreased by about 20 K when compared to the first case in Figure 10. This confirms that radiative heat transfer is not negligible when the surface emissivity is close to 1.

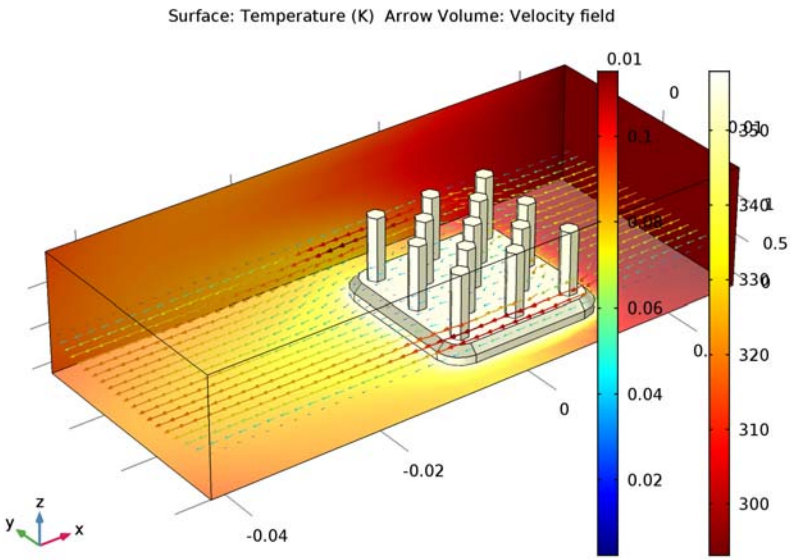
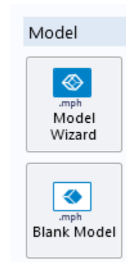


Figure 11: The effects of surface-to-surface radiation on temperature and velocity fields. The surface plot shows the temperature field on the channel walls and the heat sink surface, while the arrow plot shows the flow velocity field around the heat sink.

Model Wizard

Note: These instructions are for the user interface on Windows but apply, with minor differences, also to Linux and Mac.

- To start the software, double-click the COMSOL icon on the desktop. When the software opens, you can choose to use the



Model Wizard to create a new COMSOL model or Blank Model to create one manually. For this tutorial, click the Model Wizard button.

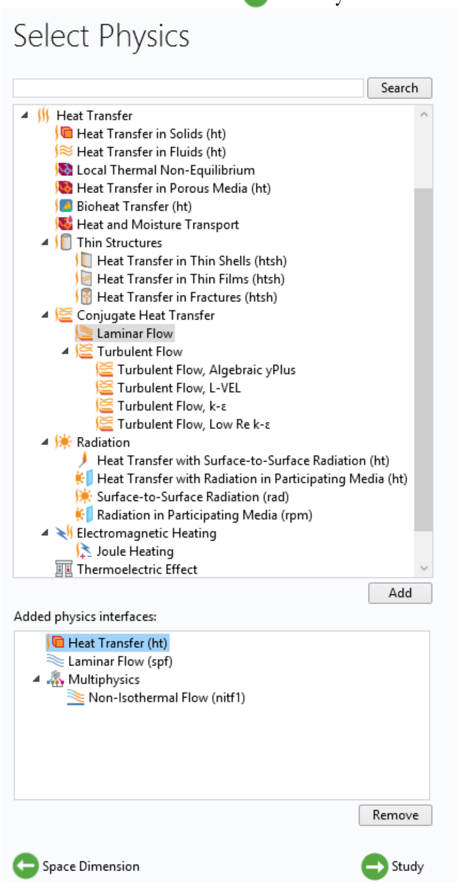
If COMSOL is already open, you can start the Model Wizard by selecting New from the File menu and then click Model Wizard .

The Model Wizard guides you through the first steps of setting up a model. The next window lets you select the dimension of the modeling space.

2 In the Select Space Dimension window click 3D .

3 In the Select Physics tree, under Heat Transfer>Conjugate Heat Transfer, click Laminar Flow .

4 Click Add and then Study to continue.



5 Under Preset Studies for Selected Physics Interfaces click Stationary .

6 Click Done .

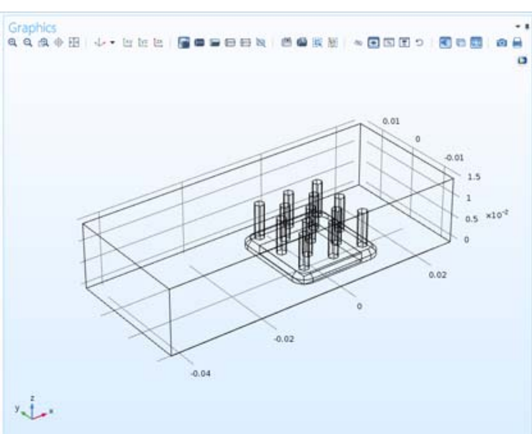
Preset Studies have solver and equation settings adapted to the selected physics interface (in this example, Conjugate Heat Transfer). A Stationary study is used in this case—there is no time-varying flow rate or heat source.

Geometry 1

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

Note: The exact location of the files used in this exercise may 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\COMSOL52a\Multiphysics\applications\Heat_Transfer_Module\Tutorials,_Forced_and_Natural_Convection\`

1. On the Geometry toolbar, click Insert Sequence.
2. Browse to the application library folder and double-click the file `heat_sink_geom_sequence.mph`. The file containing the sequence is found in `COMSOL52a\Multiphysics\applications\Heat_Transfer_Module\Tutorials,_Forced_and_Natural_Convection\`
   Double-click to add or click Open.
3. On the Geometry toolbar, click the Build All .
4. To facilitate face selection in the next steps, use the Wireframe Rendering option. Click the Wireframe Rendering button and Zoom Extents button on the Graphics toolbar.



Laminar Flow (spf)

In this part, you specify the air domain in the two physics interfaces.

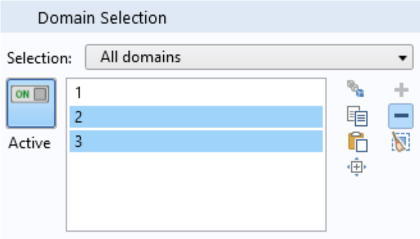
1 In the Model Builder window, under Component 1 (comp1) click Laminar Flow (spf).

2 In the Settings window for Laminar Flow, locate the Domain Selection section.

3 In the Selection list, choose `2` and `3`.

4 Click Remove from Selection.
Then domain 1should remain in the selection.

5 Click Create Selection.



6 In the Create Selection dialog box, type `Air` in the Selection name text field.

7 Click OK.

The selection for the air domain is created. Use it again in the next steps.

Heat Transfer (ht)

By default all domains are selected in the Solids feature. Add the air domain to the Fluids feature selection and it will be automatically removed (overridden) from the Solids feature selection.

Fluids 1

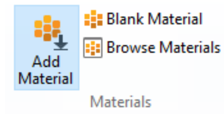
1 In the Model Builder window, under Heat Transfer (ht) click Fluids 1.

2 In the Settings window for Fluid, locate the Domain Selection section.

3 From the Selection list, choose Air.

Air

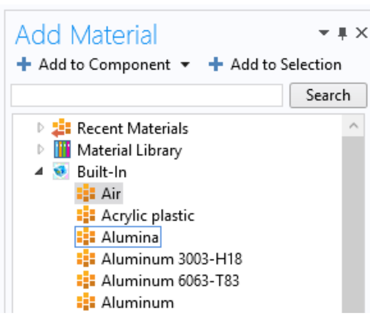
1 On the Home toolbar click Add Material .



Note: On Linux and Mac, the Home toolbar refers to the specific set of controls near the top of the Desktop.

2 In the Add Material window under Built-In, click Air .

3 Click Add to Component .



4 In the Settings window for Material, locate the Geometric Entity Selection section.

5 From the Selection list, choose Air.

Next, add the materials for the remaining domains.

Aluminum 3003-H18

1 In the Add Material window under Built-In, click Aluminum 3003-H18 .

2 In the Add Material window, click Add to Component .

3 In the Graphics window, position the mouse over the heat sink. Use the scroll wheel or arrow keys to highlight the heat sink, then click. This adds Domain 2 to the Selection list.

Silica Glass

1 Return to the Add Material window and under Built-In, select Silica glass.

2 Click Add to Component .

3 In the Graphics window, position the mouse over the chip (the small box underneath the heat sink). Use the scroll wheel or arrow keys to highlight it, then click. This adds Domain 3 to the Selection list.

Note: There are many ways to select geometric entities. To select a domain that is underneath or inside another one, first position the mouse over the desired domain. This will highlight the outer domain. You can select the desired domain from this position using the scroll wheel of a mouse or the arrow keys, then clicking once the correct domain is highlighted. This adds the domain to the Selection window. For more information about selecting geometric entities in the Graphics window, see the *COMSOL Multiphysics Reference Manual*.

4 On the Home toolbar click Add Material to close the Add Material window.

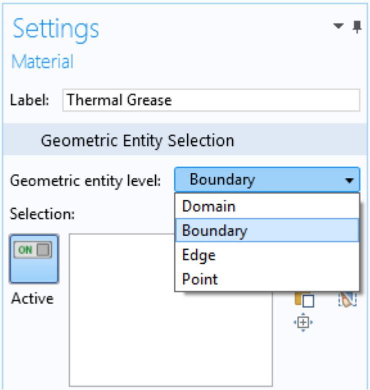
Thermal Grease

Since this material is not predefined, create a blank (new) material and define the material properties.

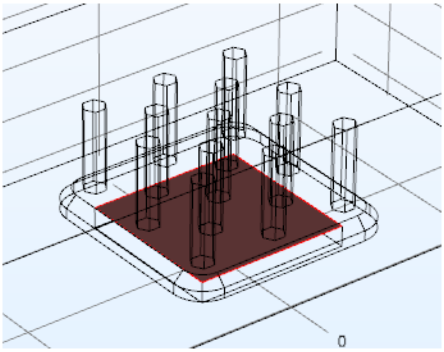
1 On the Material toolbar click Blank Material .

2 In the Settings window for Material, type `Thermal Grease` in the Label text field.

3 Locate the Geometric Entity Selection section. From the Geometric entity level list, choose Boundary.

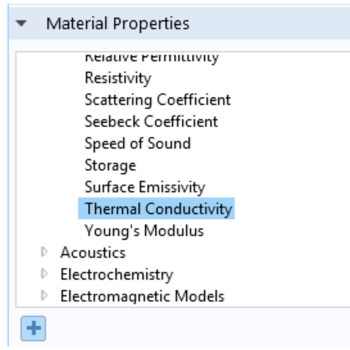


4 Select Boundary 34, which represents the interface between the heat sink and the chip.

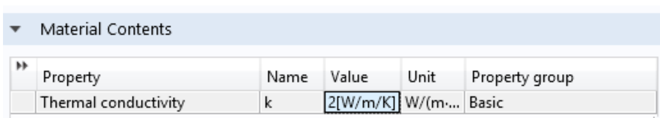


5 Click to expand the Material properties section. In the Material properties tree under Basic Properties, click Thermal Conductivity.

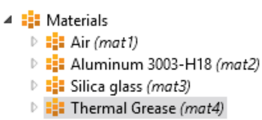
6 Click the Add to Material button.



7 Locate the Material Contents section. In the table that defines the Thermal conductivity, enter `2[W/m/K]`:



The final node sequence under Materials should match this figure.



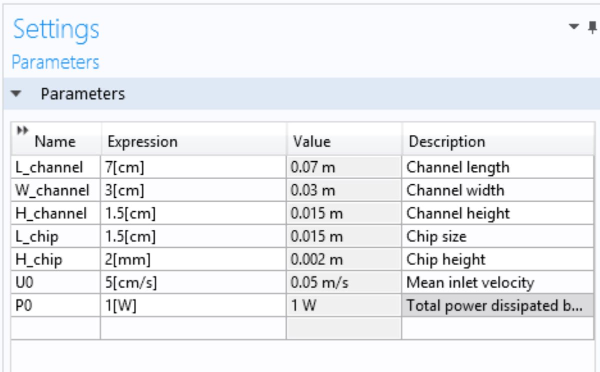
Parameters

1 On the Home toolbar click Parameters.

2 In the Settings window for Parameters, locate the Parameters section. In the table, add the following to the existing list:

NAME	EXPRESSION	DESCRIPTION
`U0`	`5[cm/s]`	Mean inlet velocity
`P0`	`1[W]`	Total power dissipated by the electronics package

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



Settings
Parameters
Parameters

Name	Expression	Value	Description
L_channel	7[cm]	0.07 m	Channel length
W_channel	3[cm]	0.03 m	Channel width
H_channel	1.5[cm]	0.015 m	Channel height
L_chip	1.5[cm]	0.015 m	Chip size
H_chip	2[mm]	0.002 m	Chip height
U0	5[cm/s]	0.05 m/s	Mean inlet velocity
P0	1[W]	1 W	Total power dissipated b...

Laminar Flow (spf)

Now the physical properties of the model are defined.

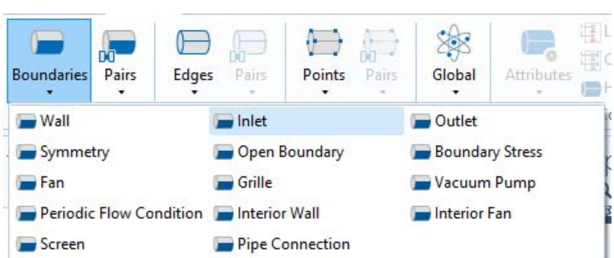
On the Physics toolbar, click Heat Transfer (ht) and choose Laminar Flow (spf).

Laminar Flow

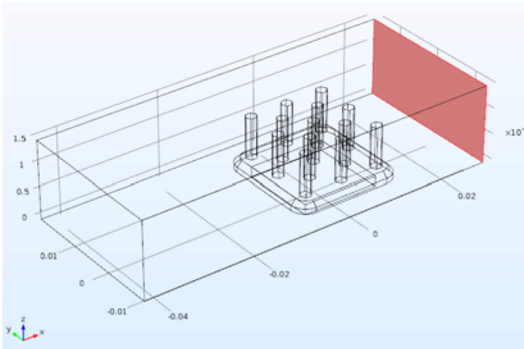
Inlet 1

The no-slip condition is the default boundary condition for the fluid. Define the inlet and outlet conditions as described below.

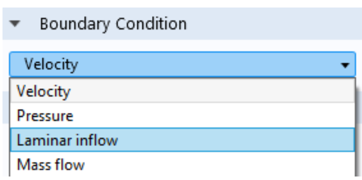
- On the Physics toolbar click Boundaries choose Inlet .



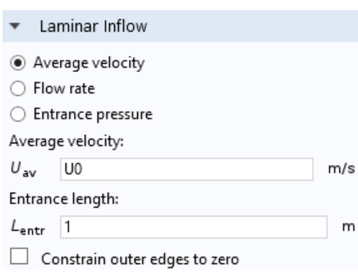
2 Select the inlet, Boundary `121`. Verify that it appears in the Settings window for Inlet under Boundary Selection



3 In the Boundary condition list select Laminar inflow.



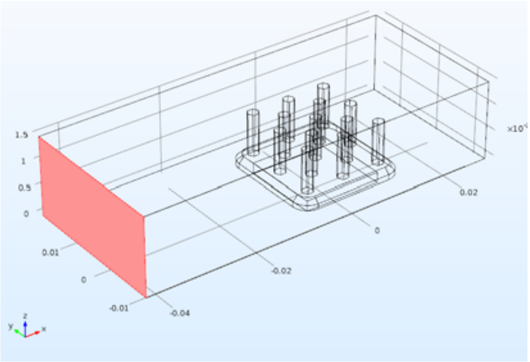
4 Under Laminar Inflow, in the U_{av} text field, enter `U0`.



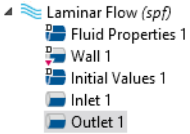
Outlet 1

1 On the Physics toolbar click Boundaries and choose Outlet .

2 Select the outlet, Boundary 1. Verify that it appears in the Settings window for Outlet under Boundary Selection.



The node sequence for the Laminar Flow part in the Model Builder should match this figure so far.



Note: The 'D' in the upper-left corner of a node means it is a default node.

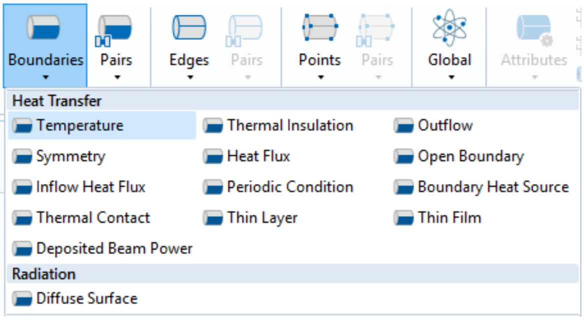
Heat Transfer (ht)

On the Physics toolbar, click Laminar Flow (spf) and choose Heat Transfer (ht).

Thermal insulation is the default boundary condition for the temperature. Define the inlet temperature and the outlet condition as described below.

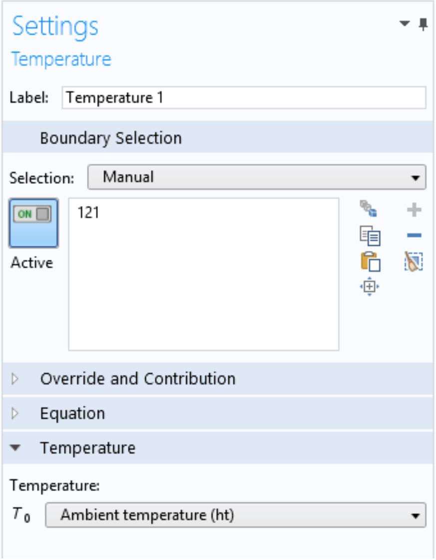
Temperature 1

1 On the Physics toolbar click Boundaries and choose Temperature.



2 Select the inlet, Boundary 121. Verify that it appears in the Settings window for Temperature under Boundary Selection.

3 Under the Temperature section from the T_0 list, choose Ambient temperature (ht).



The ambient temperature is defined in the main node of the Heat Transfer (ht) interface. Its default value is 293.15 K which corresponds to the inlet temperature used in this model. It is possible to edit the ambient temperature value or to define it using the Meteorological data option which gives access to climate data from more than 6000 stations in the world.

Outflow 1

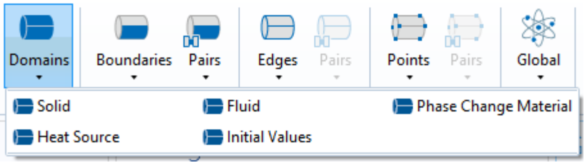
1 On the Physics toolbar click Boundaries and choose Outflow.

Select the outlet, Boundary 1. Verify that it appears in the Settings window for Outflow under Boundary Selection.

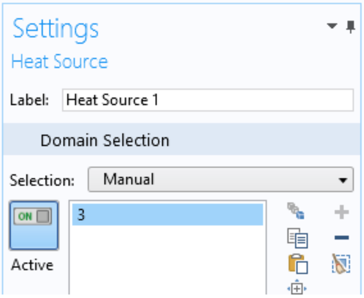
Next use the power parameter, P0, to define the total heat source in the electronics package.

Heat Source 1

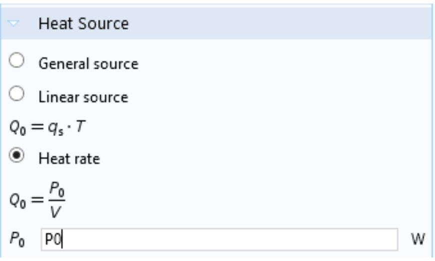
1 On the Physics toolbar click Domains and choose Heat Source.



2 Select Domain 3 corresponding to the electronics packaging. Verify that it appears in the Settings window for Heat Source under Domain Selection.



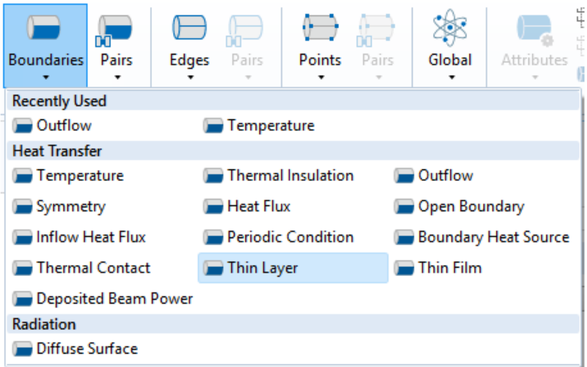
3 Under the Heat Source section, select Heat rate. In the P_0 text field, enter P0.



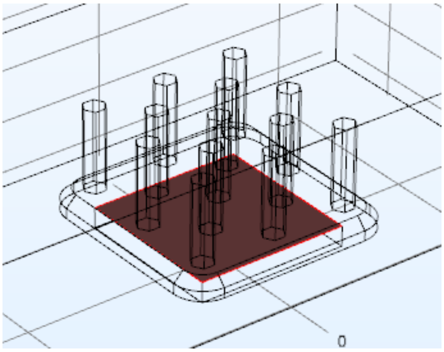
Finally, add the thin thermal grease layer.

Thin Layer 1

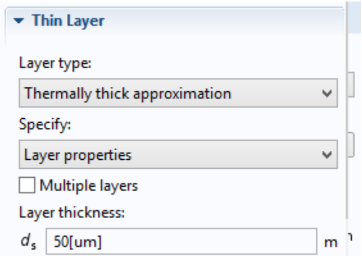
1 On the Physics toolbar click Boundaries and choose Thin Layer.



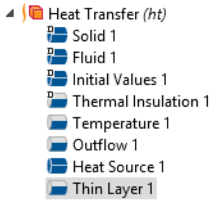
2 Select the interface between the electronics packaging and the heat sink, Boundary `34`. Verify that it appears in the Settings window for Thin Layer under Boundary Selection.



3 Under Thin Layer in the d_s text field, enter `50[um]`.

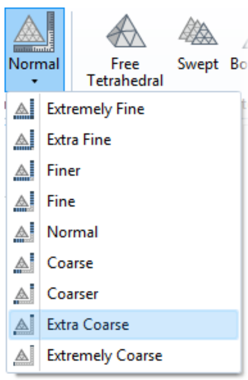


The node sequence for the Heat Transfer part in the Model Builder should match the figure.



Mesh 1

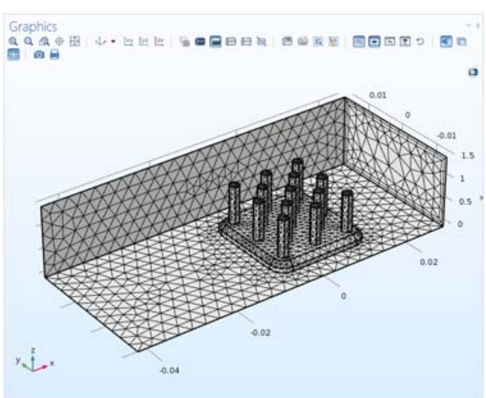
Free Tetrahedral 1 and Size



1 On the Mesh toolbar click Normal. Select Extra Coarse to replace the setting for the Mesh Size.
2 Click the Build Mesh button.

To get a better view of the mesh, hide some of the boundaries.

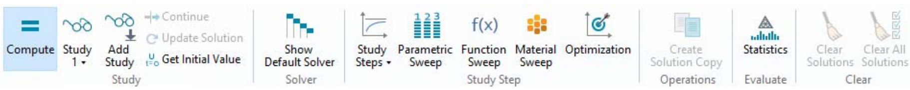
3 In the Graphics window click the Click and Hide button and then select Boundaries 1 (left), 2 (front), and 4 (top). The mesh is displayed as shown in the figure below.



To achieve more accurate 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 On the Study toolbar click Compute .



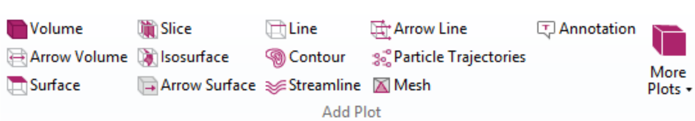
The computation takes about 2 minutes and requires 2 GB of memory.

Four default plots are generated automatically. The first one shows the temperature on the wall boundaries, the second the isothermal surfaces, the third one the velocity magnitude on five parallel slices, and the last one shows the pressure field on the wall boundaries. Next, edit the Temperature plot by adding an arrow plot to visualize the velocity field.

Results

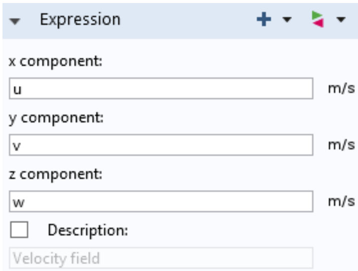
Temperature (ht)

1 In the Model Builder select the Temperature (ht) plot group. On the Temperature (ht) toolbar, click Arrow Volume .



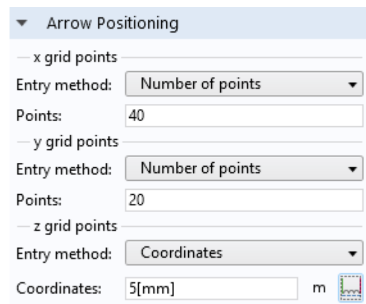
2 In the Settings window for Arrow Volume, click Replace Expression in the upper-right corner of the Expression section. From the list, select

Model>Component 1>Laminar Flow>Velocity and pressure>u,v,w - Velocity field. Double-click or press enter to add the expression.



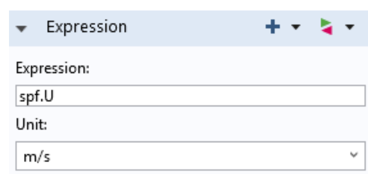
3 Under Arrow Positioning:

- In the x grid points subsection, enter `40` in the Points text field.
- In the y grid points subsection, enter `20` in the Points text field.
- In the z grid points subsection, from the Entry method list, select Coordinates.
- In the Coordinates text field, enter `5e-3` or `5[mm]`.



4 On the Temperature (ht) toolbar, click Color Expression.

5 In the upper-right corner of the Expression section, click Replace Expression.



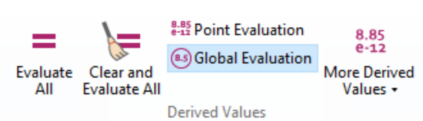
From the list, select Model>Component 1>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude or type `spf.U`.

6 Click the Plot button.

The plot in Figure 10 is displayed in the Graphics window.

To check the energy balance for a stationary model, compare the net energy flux through the system exterior boundaries (given by the variable `ht.ntefluxInt`) with the sum of the total heat sources (given by the variable `ht.QInt`).

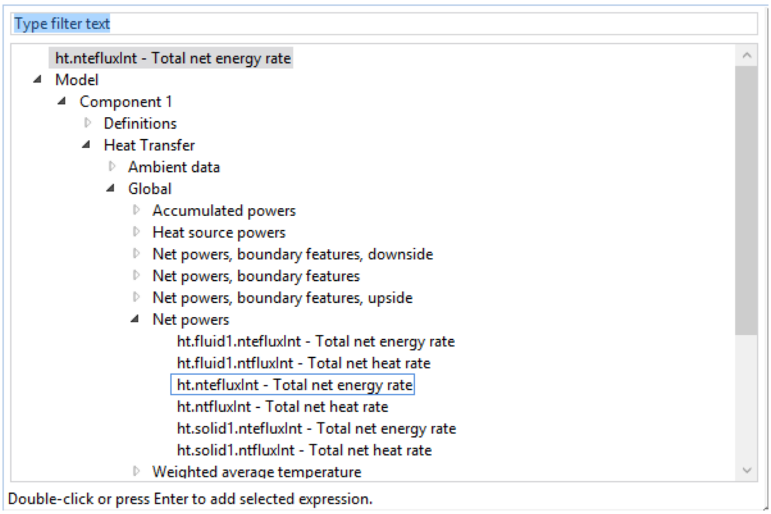
7 On the Results toolbar, click Global Evaluation.



8 In the Label text field, type `Energy Balance`.

9 Go to the Settings window for Global Evaluation. In the upper-right corner of the Expression section, click Add Expression .

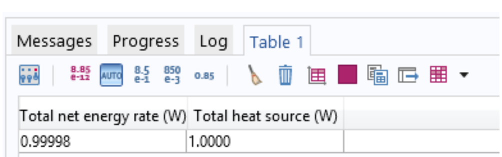
10 From the list, select Component 1>Heat Transfer> Global>Net powers>ht.ntefluxInt - Total net energy rate or type `ht.ntefluxInt`.



11 Click again on Add Expression (Ctrl+Space) .From the list select Component 1>Heat Transfer>Global > Heat source powers >ht.QInt - Total heat source or type `ht.QInt`.

12 In the Settings window for Global Evaluation, click Evaluate ..

You you can verify that the two values match and are close to 1 W in the Table 1 tab.



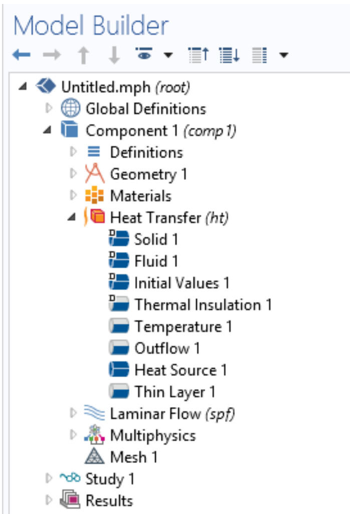
Adding Surface-to-Surface Radiation Effects

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

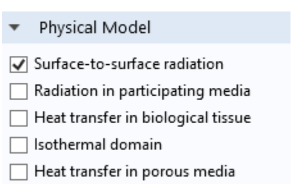
Heat Transfer (ht)

You can continue using the model built so far (in this case skip step 1 below), or you can open the model `heat_sink_surface_radiation` from the Application Libraries window.

1 In the Model Builder click Heat Transfer (ht).



2 Now, to activate surface-to-surface radiation features in the model, locate the Settings window for Heat Transfer and, under the Physical Model section, click to select the Surface-to-surface radiation check box.

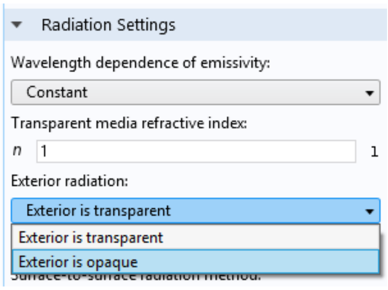


By default the radiation direction is controlled by the opacity of the domains. The solid parts are defined as opaque by default while the fluid parts are transparent by default. You can change these settings by modifying or adding an Opaque subnode under the Heat Transfer in Solids and Heat Transfer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

When the Diffuse Surface boundary condition defines the radiation direction as Opacity Controlled (default setting) the selected boundaries should be located between an opaque and a transparent domain. The exterior is defined as transparent by default. Change the default setting to make exterior opaque and have the radiation direction automatically defined on the channel walls.

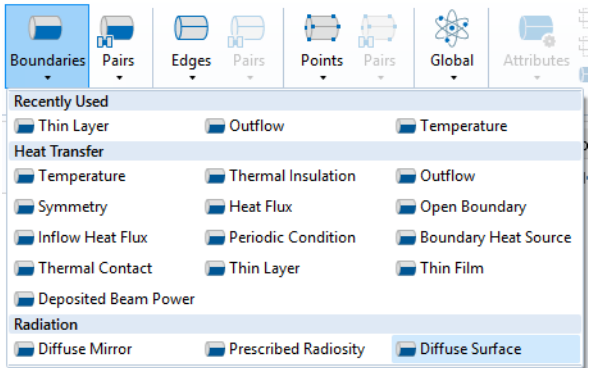
3 Once the Surface-to-surface check box is selected a Radiation Settings section appears. Locate the Exterior radiation setting.

4 From the Exterior radiation list, choose Exterior is opaque.



Diffuse Surface 1

1 On the Physics toolbar click Boundaries and under Radiation choose Diffuse Surface .

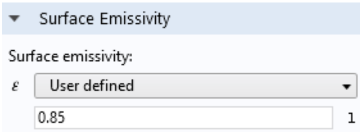


2 Select Boundaries 2–7,9–30,36–109, and 111–120.

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 text field. For more information about selecting geometric entities in the Graphics window, see the *COMSOL Multiphysics Reference Manual*.

3 In the Settings window for Diffuse Surface, under the Surface Emissivity section, from the ε list, choose User defined.

4 Enter `0.85` in the Surface emissivity text field.



Add Study

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 On the Home toolbar click Add Study.
2 The Add Study window opens. Under Preset Studies select Stationary and click Add Study.
3 To hide the Add Study window, on the Home toolbar click Add Study again.

Study 2

Step 1: Stationary

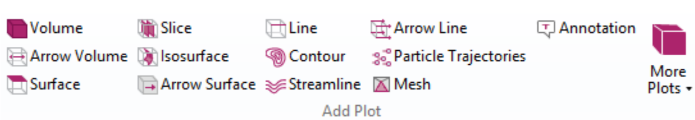
1 On the Home toolbar click Compute.

Results

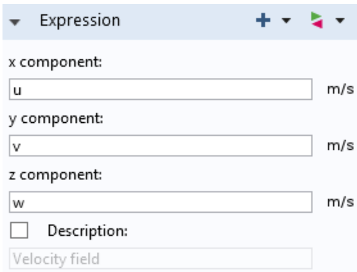
The same default plots are generated as in the study 1. Edit the temperature plot to compare the results.

Temperature (ht) 1

1 In the Model Builder, under Results, click Temperature (ht) 1. On the Temperature (ht) 1 toolbar click Arrow Volume.

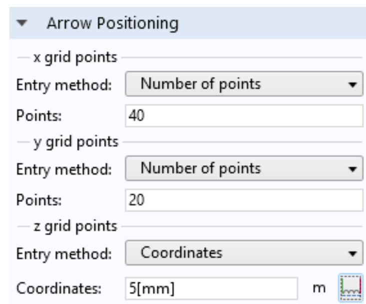


2 In the Settings windows for Arrow Volume, click Replace Expression in the upper-right corner of the Expression section. From the list, select Model>Component 1>Laminar Flow>Velocity and pressure>u,v,w - Velocity field. Double-click or press enter to add the expression.



3 Under Arrow Positioning:

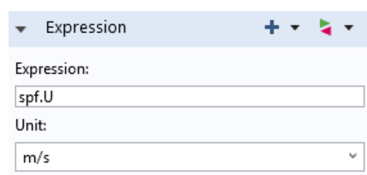
- In the x grid points subsection, enter `40` in the Points text field.
- In the y grid points subsection, enter `20` in the Points text field.
- In the z grid points subsection, from the Entry method list, select Coordinates.
- In the Coordinates text field, enter `5e-3` or `5[mm]`.



4 On the Temperature (ht) 1 toolbar, click Color Expression .

5 In the upper-right corner of the Expression section, click Replace Expression .

From the list, select Model>Component 1>Laminar Flow>Velocity and pressure>spf.U - Velocity magnitude or type `spf.U`.



6 Click the Plot button.

The plot that displays should look the same as below and Figure 11.