Heat Transfer Module

User's Guide

5.2a

COMSOL

Heat Transfer Module User's Guide

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: CM020801

Contents

Chapter 1: Introduction

Chapter 2: Notations

Chapter 3: Modeling with the Heat Transfer Module

Chapter 4: Theory for the Heat Transfer Module

Chapter 5: The Heat Transfer Module Interfaces

Chapter 7: The Moisture Transport Features

Chapter 8: Multiphysics Interfaces

Chapter 9: Multiphysics Couplings

1

Introduction

This guide describes the Heat Transfer Module, an optional package that extends the COMSOL Multiphysics® modeling environment with customized physics interfaces for the analysis of heat transfer.

This chapter introduces you to the capabilities of this module. A summary of the physics interfaces and where you can find documentation and model examples is also included. The last section is a brief overview with links to each chapter in this guide.

- About the Heat Transfer Module
- Overview of the User's Guide

About the Heat Transfer Module

In this section:

- Why Heat Transfer is Important to Modeling
- How the Heat Transfer Module Improves Your Modeling
- The Heat Transfer Module Physics Interface Guide
- Common Physics Interface and Feature Settings and Nodes
- The Heat Transfer Module Study Capabilities
- The Liquids and Gases Materials Database
- Where Do I Access the Documentation and Application Libraries?

The Physics Interfaces and Building a COMSOL Multiphysics Model in the *COMSOL Multiphysics Reference Manual*.

Why Heat Transfer is Important to Modeling

The Heat Transfer Module is an optional package that extends the COMSOL Multiphysics modeling environment with customized physics interfaces and functionality optimized for the analysis of heat transfer. It is developed for a wide audience including researchers, developers, teachers, and students. To assist users at all levels of expertise, this module comes with a library of ready-to-run examples that appear in the companion Heat Transfer Module Applications Libraries.

Heat transfer is involved in almost every kind of physical process, and can in fact be the limiting factor for many processes. Therefore, its study is of vital importance, and the need for powerful heat transfer analysis tools is virtually universal. Furthermore, heat transfer often appears together with, or as a result of, other physical phenomena.

The modeling of heat transfer effects has become increasingly important in product design including areas such as electronics, automotive, and medical industries. Computer simulation has allowed engineers and researchers to optimize process efficiency and explore new designs, while at the same time reducing the need for costly experimental trials.

How the Heat Transfer Module Improves Your Modeling

The Heat Transfer Module has been developed to greatly expand upon the base capabilities available in COMSOL Multiphysics. The module supports all fundamental mechanisms including conductive, convective, and radiative heat transfer. Using the physics interfaces in this module along with the inherent multiphysics capabilities of COMSOL Multiphysics, you can model a temperature field in parallel with other physics — a versatile combination increasing the accuracy and predicting power of your models.

This book introduces the basic modeling process. The different physics interfaces are described and the modeling strategy for various cases is discussed. These sections cover different combinations of conductive, convective, and radiative heat transfer. This guide also reviews special modeling techniques for thin layers, thin shells, participating media, and out-of-plane heat transfer. Throughout the guide the topics and examples increase in complexity by combining several heat transfer mechanisms and also by coupling these to physics interfaces describing fluid flow — conjugate heat transfer.

Another source of information is the Heat Transfer Module Applications Libraries, a set of fully-documented examples that is divided into broadly defined application areas where heat transfer plays an important role — electronics and power systems, processing and manufacturing, and medical technology — and includes tutorial and verification models.

Most of the examples involve multiple heat transfer mechanisms and are often coupled to other physical phenomena, for example, fluid dynamics, moisture transport, or electromagnetics. The authors developed several state-of-the art examples by reproducing examples that have appeared in international scientific journals. See Where Do I Access the Documentation and Application Libraries?.

The Heat Transfer Module Physics Interface Guide

The table below lists all the physics interfaces specifically available with this module. Having this module also enhances these COMSOL Multiphysics basic interfaces: Heat Transfer in Fluids, Heat Transfer in Solids, Joule Heating, and the Single-Phase Flow, Laminar interface.

If you have a Subsurface Flow Module combined with the Heat Transfer Module, this module also enhances the Heat Transfer in Porous Media interface.

The Non-Isothermal Flow, Laminar Flow (nitf) and Non-Isothermal Flow, Turbulent Flow (nitf) interfaces found under the **Fluid Flow>Non-Isothermal Flow** branch are identical to the Conjugate Heat Transfer interfaces (Laminar Flow and Turbulent Flow) found under the **Heat Transfer>Conjugate Heat Transfer** branch. The difference is that Fluid is the default domain node for the Non-Isothermal Flow interfaces.

In the *COMSOL Multiphysics Reference Manual:*

- Studies and Solvers
- The Physics Interfaces
- For a list of all the core physics interfaces included with a COMSOL Multiphysics license, see Physics Interface Guide.

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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
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
Non-Isothermal Flow				
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
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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Bioheat Transfer		ht	all dimensions	stationary; time dependent
Thermoelectric Effect(2)		—	all dimensions	stationary; time dependent
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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Heat Transfer with Radiation in Participating Media		ht	3D, 2D	stationary; time dependent
Surface-to-Surface Radiation		rad	all dimensions	stationary; time dependent
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.				

Common Physics Interface and Feature Settings and Nodes

There are several common settings and sections available for the physics interfaces and feature nodes. Some of these sections also have similar settings or are implemented in the same way no matter the physics interface or feature being used. There are also some physics feature nodes that display in COMSOL Multiphysics.

In each module's documentation, only unique or extra information is included; standard information and procedures are centralized in the *COMSOL Multiphysics Reference Manual*.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The Heat Transfer Module Study Capabilities

Table 1-1 lists the Preset Studies available for the physics interfaces most relevant to this module.

Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

TABLE 1-1: HEAT TRANSFER MODULE DEPENDENT VARIABLES AND PRESET STUDY AVAILABILITY

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	PRESET STUDIES[1]			
			STATIONARY	TIME DEPENDENT	STATIONARY WITH INITIALIZATION	TRANSIENT WITH INITIALIZATION
FLUID FLOW>SINGLE-PHASE FLOW						
Laminar Flow	spf	u, p	√	√		
Turbulent Flow, k-ε	spf	u, p, k, ep	√	√		
Turbulent Flow, Low Re k-ε	spf	u, p, k, ep, G			√	√
FLUID FLOW>NON-ISOTHERMAL FLOW						
Laminar Flow(2,4)	—	u, p, T	√	√		
Turbulent Flow, k-ε(2,4)	—	u, p, k, ep, T	√	√		
Turbulent Flow, Low Re k-ε(2,4)	—	u, p, k, ep, G, T			√	√
HEAT TRANSFER						
Heat Transfer in Solids(2)	ht	T	√	√		
Heat Transfer in Fluids(2)	ht	T	√	√		
Heat Transfer in Porous Media(2)	ht	T	√	√		
Bioheat Transfer(2)	ht	T	√	√		
Heat and Moisture Transport	—	ϕ	√	√		
Thermoelectric Effect(4)	—	T	√	√		

TABLE 1-1: HEAT TRANSFER MODULE DEPENDENT VARIABLES AND PRESET STUDY AVAILABILITY

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	PRESET STUDIES[1]			
			STATIONARY	TIME DEPENDENT	STATIONARY WITH INITIALIZATION	TRANSIENT WITH INITIALIZATION
HEAT TRANSFER>THIN STRUCURES						
Heat Transfer in Thin Shells(3)	htsh	T	√	√		
Heat Transfer in Thin Films(3)	htsh	T	√	√		
Heat Transfer in Fractures(3)	htsh	T	√	√		
HEAT TRANSFER>CONJUGATE HEAT TRANSFER						
Laminar Flow(2,4)	—	u, p, T	√	√		
Turbulent Flow, k-ε(2,4)	—	u, p, k, ep, T	√	√		
Turbulent Flow, Low Re k-ε(2,4)	—	u, p, k, ep, G, T			√	√
HEAT TRANSFER>RADIATION						
Heat Transfer with Surface-to-Surface Radiation(2)	ht	T, J	√	√		
Heat Transfer with Radiation in Participating Media(2)	ht	T, I (radiative intensity)	√	√		
Surface-to-Surface Radiation	rad	J	√	√		
Radiation in Participating Media	rpm	I (radiative intensity)	√	√		
HEAT TRANSFER>ELECTROMAGNETIC HEATING						
Joule Heating(2,4)	—	T, V	√	√		

[1] Custom studies are also available based on the physics interface.

[2] For these physics interfaces, it is possible to enable surface to surface radiation and radiation in participating media. In these cases, J and I are dependent variables.

[3] For these physics interfaces, it is possible to enable surface to surface radiation. In this case, J is a dependent variable.

[4] Multiphysics interfaces

The Liquids and Gases Materials Database

The Heat Transfer Module includes an additional Liquids and Gases material database with temperature-dependent fluid dynamic and thermal properties.

For detailed information about materials and the Liquids and Gases Material Database, see Materials in the *COMSOL Multiphysics Reference Manual*.

Where Do I Access the Documentation and Application Libraries?

A number of Internet resources have more information about COMSOL, including licensing and technical information. The electronic documentation, topic-based (or context-based) help, and the application libraries are all accessed through the COMSOL Desktop.

If you are reading the documentation as a PDF file on your computer, the blue links do not work to open an application or content referenced in a different guide. However, if you are using the Help system in COMSOL Multiphysics, these links work to open other modules, application examples, and documentation sets.

THE DOCUMENTATION AND ONLINE HELP

The *COMSOL Multiphysics Reference Manual* describes the core physics interfaces and functionality included with the COMSOL Multiphysics license. This book also has instructions about how to use COMSOL and how to access the electronic Documentation and Help content.

Opening Topic-Based Help

The Help window is useful as it is connected to many of the features on the GUI. To learn more about a node in the Model Builder, or a window on the Desktop, click to highlight a node or window, then press F1 to open the Help window, which then displays information about that feature (or click a node in the Model Builder followed by the **Help** button (?). This is called *topic-based* (or *context) help*.

Win

To open the **Help** window:

- In the **Model Builder**, **Application Builder**, or **Physics Builder** click a node or window and then press F1.
- On any toolbar (for example, **Home**, **Definitions**, or **Geometry**), hover the mouse over a button (for example, **Add Physics** or **Build All**) and then press F1.
- From the **File** menu, click **Help** (?).
- In the upper-right corner of the COMSOL Desktop, click the **Help** (?) button.

Mac

Linux

To open the **Help** window:

- In the **Model Builder** or **Physics Builder** click a node or window and then press F1.
- On the main toolbar, click the **Help** (?) button.
- From the main menu, select **Help>Help**.

Opening the Documentation Window

Win

To open the **Documentation** window:

- Press Ctrl+F1.
- From the **File** menu select **Help>Documentation** ( ).

Mac

Linux

To open the **Documentation** window:

- Press Ctrl+F1.
- On the main toolbar, click the **Documentation** ( ) button.
- From the main menu, select **Help>Documentation**.

THE APPLICATION LIBRARIES WINDOW

Each application includes documentation with the theoretical background and step-by-step instructions to create a model application. The applications are available in COMSOL as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual applications as a template for your own modeling and applications. In most models, SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

Once the Application Libraries window is opened, you can search by name or browse under a module folder name. Click to view a summary of the application and its properties, including options to open it or a PDF document.

The Application Libraries Window in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window

To open the **Application Libraries** window ( ):

Win

- From the **Home** toolbar, **Windows** menu, click ( ) **Applications Libraries**.
- From the **File** menu select **Application Libraries**.

To include the latest versions of model examples, from the **File>Help** menu, select ( ) **Update COMSOL Application Library**.

Mac

Linux

Select **Application Libraries** from the main **File>** or **Windows>** menus.

To include the latest versions of model examples, from the **Help** menu select ( ) **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

For general product information, contact COMSOL at info@comsol.com.

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and case number is sent to you by email.

COMSOL WEBSITES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
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

Overview of the User's Guide

This *Heat Transfer Module User's Guide* gets you started with modeling heat transfer using COMSOL Multiphysics. The information in this guide is specific to this module. Instructions on how to use COMSOL in general are included with the *COMSOL Multiphysics Reference Manual*.

As detailed in the section Where Do I Access the Documentation and Application Libraries? this information can also be searched from the COMSOL Multiphysics **Help** menu in the COMSOL Desktop.

Table of Contents, Notations and Index

To help you navigate through this guide, see the Contents, Notations, and Index.

Modeling with the Heat Transfer Module

The Modeling with the Heat Transfer Module chapter includes the following topics:

- Heat Transfer Variables
- Moisture Transport Variables
- Using the Boundary Conditions for the Heat Transfer Interfaces
- Handling Frames in Heat Transfer
- Heat Transfer Consistent and Inconsistent Stabilization Methods
- Heat Transfer and Fluid Flow Coupling
- Boundary Wall Temperature
- Solver Settings
- Plotting Results in Thin Layers Extra Dimensions
- Using Ambient Data

Theory for the Heat Transfer Module

The Theory for the Heat Transfer Module chapter includes the theory related to the heat transfer and moisture transport interfaces and multiphysics interfaces, and also to some nodes.

After the establishment of the heat balance equation from the energy conservation laws in Foundations of the General Heat Transfer Equation, the various versions of the heat equation solved in COMSOL Multiphysics are presented in the following sections:

- Theory for Heat Transfer in Solids
- Theory for Heat Transfer in Fluids
- Theory for Bioheat Transfer
- Theory for Heat Transfer in Porous Media
- Theory for Heat Transfer with Phase Change
- Theory for Lumped Isothermal Domain
- Theory for Heat Transfer in Thin Structures
- Theory for Surface-to-Surface Radiation
- Theory for Radiation in Participating Media
- Theory for Moisture Transport

Then the theory related to multiphysics interfaces is described in Theory for the Heat Transfer Multiphysics Couplings.

Finally, topics related to specific features or variables are treated in Theory for Thermal Contact, Moist Air Fluid Type, Out-of-Plane Heat Transfer, The Heat Transfer Coefficients, Equivalent Thermal Conductivity Correlations, Temperature Dependence of Surface Tension, Heat Flux and Heat Balance, and Frames for the Heat Transfer Equations.

THE HEAT TRANSFER MODULE INTERFACES

The Heat Transfer Module Interfaces chapter describes the main Heat Transfer interface (ht) that forms the backbone for all the fundamental interfaces in this module, and the other interfaces (Heat Transfer in Thin Shells (htsh), Radiation in Participating Media (rpm), Surface-to-Surface Radiation (rad), and Moisture Transport (mt)).

The Heat Transfer in Solids Interface, The Heat Transfer in Fluids Interface, and The Heat Transfer Interface discuss modeling heat transfer in solids and fluids.

The Heat Transfer in Porous Media Interface section discusses modeling heat transfer in porous media.

The Bioheat Transfer Interface section discusses modeling heat transfer within biological tissue using the Bioheat Transfer interface.

The Heat Transfer in Thin Shells Interface, The Heat Transfer in Thin Films Interface, and The Heat Transfer in Fractures Interface sections describe the physics interfaces which are suitable for solving thermal conduction, convection, and radiation problems in thin structures.

The The Heat Transfer with Surface-to-Surface Radiation Interface, The Heat Transfer with Radiation in Participating Media Interface, The Surface-To-Surface Radiation Interface, and The Radiation in Participating Media Interface sections discuss the modeling of radiative heat transfer in transparent and participating media.

Finally, The Moisture Transport Interface section describes the modeling of moisture transfer in a porous medium through vapor diffusion and capillary moisture flows.

THE HEAT TRANSFER FEATURES

The Heat Transfer Features chapter describes the Domain Features, Boundary Features, Edge Features, Point Features, and Global Features available with the Heat Transfer interfaces.

THE MOISTURE TRANSPORT FEATURES

The Moisture Transport Features chapter describes the Domain Features and Boundary Features available with the Moisture Transport interface.

THE HEAT TRANSFER MULTIPHYSICS INTERFACES

The Multiphysics Interfaces chapter describes the predefined multiphysics interfaces.

The Thermoelectric Effect Interface section describes the predefined multiphysics interface used to model the Peltier-Seebeck-Thomson effect.

The Non-Isothermal Flow and Conjugate Heat Transfer Interfaces chapter describes the multiphysics versions of both the Non-Isothermal Flow Laminar Flow and Turbulent Flow interfaces found under the Fluid Flow branch, which are identical to the Conjugate Heat Transfer interfaces. Each section describes the applicable physics interfaces in detail and concludes with the underlying theory.

The Local Thermal Non-Equilibrium Interface section describes the predefined multiphysics interface used to model heat transfer in porous media when there is no thermal equilibrium between porous and fluid phases.

The Heat and Moisture Transport Interface section describes the predefined multiphysics interface used to model coupled heat and moisture transport in building materials, by taking into account heat and moisture storage, latent heat effects, and liquid and convective transport of moisture.

The Heat Transfer Multiphysics Couplings

The Multiphysics Couplings chapter describes the Domain Multiphysics Couplings and the Boundary Multiphysics Couplings available with the predefined multiphysics interfaces.

2

Notations

This section introduces the notations used in the remaining of the guide. The notations are listed by alphabetical order and grouped in two tables, for Latin and Greek symbols.

For each entry the SI unit and a short description are given.

LATIN SYMBOLS

NOTATION	SI UNIT	DESCRIPTION
A	1/s	Frequency factor, damage integral analysis
A	m^2	Total boundaries area
$a_1,..., a_{12}$	dimensionless	Legendre coefficients
A_c	m^2	Cross sectional area of domain
A_l	m^2	Cross sectional area of thin rod
a_{sf}	1/m	Specific surface area
B_i	m	Spectral band i
b	dimensionless	Thermal conductivity supplement
c	mol/m^3	Concentration
c	kg/m^3	Concentration
$C_{p,\,b}$	J/(kg·K)	Specific heat capacity at constant pressure, blood
$C_{p,\,l}$	J/(kg·K)	Specific heat capacity at constant pressure of thin rod
C_{μ}	dimensionless	Turbulence modeling constant
C_p	J/(kg·K)	Specific heat capacity at constant pressure
$C_{p,\,a}$	J/(kg·K)	Specific heat capacity at constant pressure, dry air
$C_{p,\,d}$	J/(kg·K)	Specific heat capacity at constant pressure, damaged tissue
$C_{p,\,f}$	J/(kg·K)	Specific heat capacity at constant pressure, fluid phase
$C_{p,\,fr}$	J/(kg·K)	Specific heat capacity at constant pressure of solid material in fracture
$C_{p,\,g}$	J/(kg·K)	Specific heat capacity at constant pressure of immobile fluid in porous media
$C_{p,\,gi}$	J/(kg·K)	Specific heat capacity at constant pressure of immobile fluid i in porous media
$C_{p,\,m}$	kg/m^3	Mixture specific heat capacity at constant pressure (moist air)
$C_{p,\,p}$	J/(kg·K)	Specific heat capacity at constant pressure of solid material in porous media
$C_{p,\,pi}$	J/(kg·K)	Specific heat capacity at constant pressure of solid material i in porous media
$C_{p,\,s}$	J/(kg·K)	Specific heat capacity at constant pressure, solid phase
$C_{p,\,s}$	J/(kg·K)	Specific heat capacity at constant pressure of dry solid
$C_{p,\,v}$	J/(kg·K)	Specific heat capacity at constant pressure, water vapor
$C_{p,\,s}$	J/(kg·K)	Specific heat capacity at constant pressure of thin layer

NOTATION	SI UNIT	DESCRIPTION
c_{sat}	mol/m^3	Saturation concentration of water vapor
$C_{p,\,\mathrm{s}i}$	J/(kg·K)	Specific heat capacity at constant pressure of layer i in thin layer
$C_{p,\,\mathrm{w}}$	J/(kg·K)	Specific heat capacity at constant pressure of water
c_{v}	mol/m^3	Water vapor concentration
d	m	Average particle diameter
D	m	Cylinder diameter (heat transfer coefficient)
D	m	Cylinder or sphere diameter (heat transfer coefficient)
D	W/(m·K)	Dispersion tensor
D	m	Parallel-plate gap average gas particle diameter
D	1/s	Strain-rate tensor
Da	dimensionless	Darcy number
ΔE	J/mol	Activation energy, damage integral analysis
d_{f}	m	Thickness of thin film
d_{fr}	m	Thickness of fracture
D_{P1}	m^2/s	P1 method diffusion coefficient
d_{s}	m	Thickness of shell or thin layer
$d_{\mathrm{s}i}$	m	Thickness of layer i in thin layer
D_{w}	m^2/s	Moisture diffusivity
d_z	m	Thickness of domain in the out-of-plane direction
DPT_{amb}	K	Ambient dew point temperature
$\mathbf{e}$	m	Beam orientation
E	J/kg	Internal energy
E	Pa	Young's modulus
E_0	J/kg	Total internal energy
E_{Ω}	J	Internal energy of a body
$e_{\mathrm{b},\,\lambda}(\lambda, T)$	W/m^2	Blackbody spectral emissive power
$e_{\mathrm{b}}(T)$	W/m^2	Blackbody total emissive power
E_{contact}	Pa	Effective contact interface Young's modulus
E_{d}	Pa	Young's modulus, down contact surface
E_{k}	J/kg	Kinetic energy
ep	m^2/s^3	Turbulent dissipation rate

NOTATION	SI UNIT	DESCRIPTION
E_{p}	J/kg	Potential energy
E_{u}	Pa	Young's modulus, up contact surface
$\mathbf{e}_{\mathrm{tot}}$	W/m^2	Total energy flux
$\mathbf{F}$	N/m^3	Body force vector
F	dimensionless	Deformation gradient
F_{amb}	dimensionless	Ambient view factor
$F_{\mathrm{amb,\,d}}$	dimensionless	Ambient view factor, downside
$F_{\mathrm{amb,\,u}}$	dimensionless	Ambient view factor, upside
g	m/s^2	Acceleration of gravity
G	W/m^3	Moisture source
G	1/m	Reciprocal wall distance
G	W/m^2	Surface irradiation
G_{amb}	W/m^2	Ambient irradiation
$G_{\mathrm{amb,\,d}}$	W/m^2	Ambient irradiation, downside
$G_{\mathrm{amb,\,u}}$	W/m^2	Ambient irradiation, upside
G_{d}	W/m^2	Surface irradiation, downside
G_{ext}	W/m^2	External irradiation
$G_{\mathrm{ext,\,d}}$	W/m^2	External irradiation, downside
$G_{\mathrm{ext,\,u}}$	W/m^2	External irradiation, upside
G_{m}	W/m^2	Mutual surface irradiation
$G_{\mathrm{m,\,d}}$	W/m^2	Mutual surface irradiation, downside
$G_{\mathrm{m,\,u}}$	W/m^2	Mutual surface irradiation, upside
Gr_L	dimensionless	Grashof number associated with characteristic length L
G_{u}	W/m^2	Surface irradiation, upside
H	m	Chimney height (heat transfer coefficient)
H	J/kg	Enthalpy
h	W/(m^2·K)	Gap conductance (thermal contact)
h	W/(m^2·K)	Heat transfer coefficient
H_0	J/kg	Total enthalpy
H_{B}	Pa	Brinell hardness
h_{c}	W/(m^2·K)	Constriction conductance
H_{c}	Pa	Microhardness

NOTATION	SI UNIT	DESCRIPTION
h_d	W/(m^2·K)	Out-of-plane heat transfer coefficient, downside
H_d	J/kg	Enthalpy, downside
H_ext	J/kg	External enthalpy
h_g	W/(m^2·K)	Parallel-plate gap gas conductance
h_r	W/(m^2·K)	Radiative conductance
H_ref	J/kg	Reference enthalpy
h_sf	W/(m^2·K)	Interstitial heat transfer coefficient
h_th	W/K	Thermal conductance (isothermal domain interface)
h_u	W/(m^2·K)	Out-of-plane heat transfer coefficient, upside
H_u	J/kg	Enthalpy, upside
h_z	W/(m^2·K)	Out-of-plane heat transfer coefficient, 1D
$I(\Omega)$	W/(m^2·sr)	Radiative intensity traveling in direction Ω
$I_\mathrm{b}(T)$	W/(m^2·sr)	Blackbody radiative intensity
I_ext	W/(m^2·sr)	Incident radiative intensity
I_i	W/(m^2·sr)	Radiative intensity traveling in ith discrete direction
$\mathbf{i}_\mathrm{s}$	dimensionless	Incident radiation direction (external radiation source)
I_s	W/m^2	Solar irradiance
$I_\mathrm{s,amb}$	W/m^2	Ambient solar irradiance
$I_\mathrm{sh,amb}$	W/m^2	Clear sky noon diffuse horizontal irradiance
$I_\mathrm{sn,amb}$	W/m^2	Clear sky noon beam normal irradiance
is_x, is_y, is_z	dimensionless	Solar source direction vector components
I_wall	W/(m^2·sr)	Boundary radiative intensity
J	W/m^2	Surface radiosity
J_d	W/m^2	Surface radiosity, downside
J_u	W/m^2	Surface radiosity, upside
k	W/(m·K)	Thermal conductivity
k	J/kg	Turbulent kinetic energy (turbulent non-isothermal flow)
k_a	W/(m·K)	Dry air thermal conductivity
k_B	J/K	Stefan-Boltzmann constant
k_bnd	W/(m·K)	Thermal conductivity in shell local coordinate system
k_contact	W/(m·K)	Harmonic mean of contacting surface conductivities

NOTATION	SI UNIT	DESCRIPTION
k_d	W/(m·K)	Thermal conductivity, damaged tissue
k_disp	W/(m·K)	Dispersive thermal conductivity tensor
k_eff	W/(m·K)	Effective thermal conductivity
k_f	W/(m·K)	Thermal conductivity, fluid phase
k_fr	W/(m·K)	Thermal conductivity of solid material in fracture
k_g	W/(m·K)	Thermal conductivity of immobile fluid in porous media
k_gap	W/(m·K)	Parallel-plate gap gas thermal conductivity
$k_{\mathrm{g}i}$	W/(m·K)	Thermal conductivity of immobile fluid i in porous media
k_l	W/(m·K)	Thermal conductivity of thin rod
k_L	W/(m·K)	Thermal conductivity of mobile fluid in porous media
k_p	W/(m·K)	Thermal conductivity of solid material in porous media
$k_{\mathrm{p}i}$	W/(m·K)	Thermal conductivity of solid material i in porous media
k_R	W/(m·K)	Rosseland radiative conductivity
k_s	W/(m·K)	Thermal conductivity of thin layer
k_s	W/(m·K)	Thermal conductivity, solid phase
k_s	W/(m·K)	Thermal conductivity of dry solid
$k_{\mathrm{s}i}$	W/(m·K)	Thermal conductivity of layer i in thin layer
k_T	W/(m·K)	Turbulent thermal conductivity
k_v	W/(m·K)	Water vapor thermal conductivity
K_Ω	J	Kinetic energy of a body
L	J/kg	Latent heat
L	m	Total edge length
L	m	Wall height or plate diameter, distance or length (heat transfer coefficient)
L_v	J/kg	Latent heat of evaporation
m	kg	Mass
m_a	kg	Dry air mass
M_a	kg/mol	Dry air molar mass
m_asp	m	Asperities average slope (surface roughness)
M_g	m	Parallel-plate gap gas parameter
M_n	kg/mol	Mean molar mass
m_tot	kg	Total mass (moist air)

NOTATION	SI UNIT	DESCRIPTION
m_{v}	kg	Water vapor mass
M_{v}	kg/mol	Water vapor molar mass
n	dimensionless	Refractive index, transparent media
$\mathbf{n}$	dimensionless	Normal vector toward exterior
n_{a}	mol	Amount of dry air
n_{r}	dimensionless	Refractive index, participating media
n_{tot}	mol	Amount of moist air
Nu	dimensionless	Nusselt number
Nu_L	dimensionless	Nusselt number associated with characteristic length L
n_{v}	mol	Amount of water vapor
$\mathbf{O}$	m	Beam origin point
p	Pa	Contact pressure, pressure
P	N/m^2	First Piola-Kirchhoff stress tensor
P	V	Peltier coefficient
P_0	W	Heat rate
p_{a}	Pa	Dry air partial pressure
p_{A}	Pa	Absolute pressure
P_{b}	W	Heat rate, boundary heat source
P_{c}	m	Cross sectional perimeter of domain
p_{ext}	Pa	External absolute pressure
P_{ext}	W	Power of applied forces
p_{gap}	Pa	Parallel-plate gap gas pressure
P_{index}	dimensionless	Performance index of the discrete ordinates method
P_{l}	W	Heat rate, line heat source, deposited beam power
Pr	dimensionless	Prandtl number
p_{amb}	Pa	Ambient absolute pressure
p_{ref}	Pa	Reference pressure
Pr_{T}	dimensionless	Turbulent Prandtl number
P_{s}	W	Heat rate, layer heat source
P_{s}	W	Source power (external radiation source)
p_{sat}	Pa	Saturation pressure of water vapor
P_{str}	W	Stress power

NOTATION	SI UNIT	DESCRIPTION
p_{v}	Pa	Water vapor partial pressure
$\mathbf{q}$	$\mathrm{W/m^2}$	Conductive heat flux
$\mathbf{q}_{\mathrm{f}}$	$\mathrm{W/m^2}$	Conductive heat flux in fluid phase
$\mathbf{q}_{\mathrm{s}}$	$\mathrm{W/m^2}$	Conductive heat flux in solid phase
Q	$\mathrm{W/m^3}$	Heat source
q_0	$\mathrm{W/m^2}$	Inward heat flux
Q_0	$\mathrm{W/m^3}$	Distributed heat source
$q_{0,\,\mathrm{d}}$	$\mathrm{W/m^2}$	Out-of-plane heat flux, downside
$q_{0,\,\mathrm{s}}$	$\mathrm{W/m^2}$	Source heat flux (external radiation source)
$q_{0,\,\mathrm{u}}$	$\mathrm{W/m^2}$	Out-of-plane heat flux, upside
Q_{b}	$\mathrm{W/m^2}$	Boundary heat source
$Q_{\mathrm{b,\,tot}}$	$\mathrm{W/m^2}$	Total boundary heat source
Q_{e}	$\mathrm{W/m^3}$	Electromagnetic heat source
Q_{exch}	W	Exchanged heat source rate
Q_{f}	$\mathrm{W/m^3}$	Heat source in fluid phase
q_{geo}	W/kg	Radiogenic heating per mass
Q_{geo}	$\mathrm{W/m^3}$	Geothermal heat source
Q_{Int}	W	Total heat source over interior boundaries
Q_{l}	W/m	Line heat source
Q_{met}	$\mathrm{W/m^3}$	Metabolic heat source
Q_{p}	W	Point heat source
Q_{p}	$\mathrm{W/m^3}$	Pressure work
Q_{r}	$\mathrm{W/m^3}$	Radiative heat source term
$\mathbf{q}_{\mathrm{r}}$	$\mathrm{W/m^2}$	Radiative heat flux
$q_{\mathrm{r,\,net}}$	$\mathrm{W/m^2}$	Net radiative heat flux
$q_{\mathrm{r,\,out}}$	$\mathrm{W/m^2}$	Radiative heat flux striking the wall
q_{s}	$\mathrm{W/(m^3{\cdot}K)}$	Production/absorption coefficient
Q_{s}	$\mathrm{W/m^3}$	Heat source in solid phase
Q_{s}	$\mathrm{W/m^3}$	Thin layer heat source
q_{sf}	$\mathrm{W/(m^3{\cdot}K)}$	Interstitial convective heat transfer coefficient
$Q_{\mathrm{s}i}$	$\mathrm{W/m^3}$	Layer i heat source
Q_{ted}	$\mathrm{W/m^3}$	Thermoelastic damping

NOTATION	SI UNIT	DESCRIPTION
$\mathbf{q}_{\mathrm{tot}}$	W/m^2	Total heat flux
Q_{tot}	W/m^3	Total domain heat source
Q_{vd}	W/m^3	Viscous dissipation
r	m	Distance of the irradiated surface from the source
r	dimensionless	Heat partition coefficient (thermal friction)
R	m	Heat source radius, beam radius
R	J/(mol·K)	Universal gas constant
Ra	dimensionless	Rayleigh number
Ra_D	dimensionless	Rayleigh number associated with cylinder diameter D
Ra_L	dimensionless	Rayleigh number associated with characteristic length L
$\mathrm{Re}_{\mathrm{inf}}$	dimensionless	Reynolds number at infinity
Re_L	dimensionless	Reynolds number associated with characteristic length L
Re_{p}	dimensionless	Particle Reynolds number
r_{h}	m	Hydraulic radius
r_{l}	m	Rod radius
r_{p}	m	Average pellet radius
R_{s}	J/(kg·K)	Specific gas constant
R_{t}	K·m^2/W	Thermal resistance
$R_{\mathrm{t,\,th}}$	K/W	Absolute thermal resistance
S	N/m^2	Second Piola-Kirchhoff stress tensor
S	V/K	Seebeck coefficient
$\mathbf{S}_i$	dimensionless	Unit vector of discrete direction in space, i-th component (angular space discretization)
Sp	dimensionless	Sparrow number
T	K	Temperature
T^+	dimensionless	Dimensionless temperature
T_{amb}	K	Ambient temperature
$T_{\mathrm{amb,\,d}}$	K	Ambient temperature, downside
$T_{\mathrm{amb,\,u}}$	K	Ambient temperature, upside
T_{b}	K	Arterial blood temperature
T_{d}	K	Temperature, downside
$t_{\mathrm{d,\,c}}$	s	Damage time, cryogenic analysis

NOTATION	SI UNIT	DESCRIPTION
$T_{\mathrm{d,\,c}}$	K	Damage temperature, cryogenic analysis
$t_{\mathrm{d,\,h}}$	s	Damage time, hyperthermia analysis
$T_{\mathrm{d,\,h}}$	K	Damage temperature, hyperthermia analysis
T_{ext}	K	External temperature
$T_{\mathrm{ext,\,d}}$	K	Out-of-plane external temperature, downside
$T_{\mathrm{ext,\,u}}$	K	Out-of-plane external temperature, upside
$T_{\mathrm{ext},\,z}$	K	Out-of-plane external temperature, 1D
T_{f}	K	Temperature, fluid phase
$T_{\mathrm{n,\,c}}$	K	Temperature of necrosis, cryogenic
$T_{\mathrm{n,\,h}}$	K	Temperature of necrosis, hyperthermia
T_{pc}	K	Phase change temperature
T_{ref}	K	Reference temperature
T_{ref}	K	Strain reference temperature
T_{s}	K	Temperature, solid phase
T_{u}	K	Temperature, upside
T_{w}	K	Wall temperature
$\mathbf{u}$	m/s	Fluid velocity vector
u, v, w	m/s	Fluid velocity vector's components
$\mathbf{u}_{\mathrm{f}}$	m/s	Average linear velocity
$\mathbf{u}_{\mathrm{p}}$	m/s	Porous velocity field
$\mathbf{u}_{\mathrm{trans}}$	m/s	Translational motion velocity vector
v_{amb}	m/s	Wind velocity
V	V	Electric potential
V	m^3	Total domain volume
W	$\mathrm{W/m}^3$	Work source
w	$\mathrm{kg/m}^3$	Moisture storage function
W_{Int}	W	Work from custom volume forces
W_{diss}	W	Dissipative work from momentum equation
X_{a}	dimensionless	Molar fraction of dry air
x_{pl}	m	Position along the plate (heat transfer coefficient)
$\mathbf{x}_{\mathrm{s}}$	m	Source location (external radiation source)
X_{v}	dimensionless	Molar fraction of water vapor

NOTATION	SI UNIT	DESCRIPTION
x_{vap}	dimensionless	Moisture content
$x_{\mathrm{vap,amb}}$	dimensionless	Ambient moisture content
Y	m	Parallel-plate gap mean separation thickness

GREEK SYMBOLS

NOTATION	SI UNIT	DESCRIPTION
$(\rho C_p)_{\mathrm{eff}}$	$J/(m^3 \cdot K)$	Effective volumetric heat capacity at constant pressure
$\partial\Omega$	-	Geometry domain's boundaries
$\partial\Omega_{\mathrm{ext}}$	-	Geometry domain's exterior boundaries
$\partial\Omega_{\mathrm{int}}$	-	Geometry domain's interior boundaries
$\nabla_{\mathbf{t}}$	-	Tangential gradient operator
α	1/K	Coefficient of thermal expansion in a solid
α	dimensionless	Degree of tissue injury (Arrhenius equation)
α	dimensionless	Parallel-plate gap gas thermal accommodation parameter
α	dimensionless	Surface absorptivity
α	m^2/s	Thermal diffusivity
α_1	dimensionless	Damaged tissue indicator
α_2	dimensionless	Necrosis time indicator
α_{m}	dimensionless	Vapor mass fraction
α_p	1/K	Coefficient of thermal expansion in a fluid
β	1/m	Extinction coefficient
β	s/m	Moisture transfer coefficient
β	dimensionless	Parallel-plate gap gas property parameter
β_{R}	1/m	Rosseland mean extinction coefficient
γ	dimensionless	Ratio of specific heats
$\gamma_{T\mathrm{eq}}$	Pa/K	Psychrometer constant
ΔR	m	Size of transition zone (deposited beam power)
ΔT	K	Temperature offset for periodic condition
δ	s	Vapor permeability of still air
δ_{p}	s	Vapor permeability
δ_{w}	m	Distance between the computational fluid domain and the wall

NOTATION	SI UNIT	DESCRIPTION
ε	dimensionless	Surface emissivity
ε_d	dimensionless	Surface emissivity, downside
ε_λ	dimensionless	Surface spectral emissivity
ε_u	dimensionless	Surface emissivity, upside
ε_z	dimensionless	Surface emissivity, 1D out-of-plane radiation
θ	rad	Angle between the normal to the irradiated surface and the direction of the source
θ_d	dimensionless	Volume fraction of necrotic tissue
θ_fr	dimensionless	Volume fraction of solid material in fracture
θ_g	dimensionless	Volume fraction of immobile fluid in porous media
$\theta_{\mathrm{g}i}$	dimensionless	Volume fraction of immobile fluid i in porous media
θ_L	dimensionless	Volume fraction of mobile fluid in porous media
θ_p	dimensionless	Volume fraction of solid material in porous media
$\theta_{\mathrm{p}i}$	dimensionless	Volume fraction of solid material i in porous media
θ_s	rad	Zenith angle of the Sun
κ	1/m	Absorption coefficient (radiation)
κ	m^2	Permeability (porous media)
λ	W/(m·K)	Thermal conductivity (turbulent non-isothermal flow)
λ	m	Wavelength
Λ	m	Parallel-plate gap gas mean free path
λ_i	m	Wavelength band i endpoint
λ_{ijkl}	m	Fourth-order dispersivity tensor’s component
λ_lo	m	Longitudinal dispersivity
λ_tr	m	Transverse dispersivity
$\lambda_\mathrm{tr,\,h}$	m	Transverse horizontal dispersivity
$\lambda_\mathrm{tr,\,v}$	m	Transverse vertical dispersivity
μ	Pa·s	Dynamic viscosity
μ	dimensionless	Vapor resistance factor
μ_0	dimensionless	Scattering angle
μ_a	Pa·s	Dry air dynamic viscosity
μ_f	Pa·s	Dynamic viscosity
μ_m	Pa·s	Mixture dynamic viscosity (moist air)

NOTATION	SI UNIT	DESCRIPTION
μ_{T}	Pa·s	Turbulent dynamic viscosity
μ_{Th}	V/K	Thomson coefficient
μ_{v}	Pa·s	Water vapor dynamic viscosity
ν	dimensionless	Poisson ratio
ν_{d}	dimensionless	Poisson ratio, down contact surface
ν_{u}	dimensionless	Poisson ratio, up contact surface
ρ	kg/m^3	Density
ρ	dimensionless	Surface reflectivity
ρ_{b}	kg/m^3	Density, blood
ρ_{d}	kg/m^3	Density, damaged tissue
ρ_{f}	kg/m^3	Density, fluid phase
ρ_{fr}	kg/m^3	Density of solid material in fracture
ρ_{g}	kg/m^3	Density of immobile fluid in porous media
ρ_{geo}	kg/m^3	Geothermal density
$\rho_{\mathrm{g}i}$	kg/m^3	Density of immobile fluid i in porous media
ρ_{l}	kg/m^3	Density of thin rod
ρ_{m}	kg/m^3	Mixture density (moist air)
ρ_{p}	kg/m^3	Density of solid material in porous media
$\rho_{\mathrm{p}i}$	kg/m^3	Density of solid material i in porous media
ρ_{s}	kg/m^3	Density of thin layer
ρ_{s}	kg/m^3	Density, solid phase
ρ_{s}	kg/m^3	Density of dry solid
$\rho_{\mathrm{s}i}$	kg/m^3	Density of layer i in thin layer
σ	Pa	Cauchy stress tensor
σ	m	Standard deviation (deposited beam power)
σ	$W/(m^2 \cdot K^4)$	Stefan-Boltzmann constant
σ	N/m	Surface tension coefficient
σ_{asp}	m	Asperities average height (surface roughness)
σ_{s}	1/m	Scattering coefficient
τ	dimensionless	Optical thickness
τ	Pa	Viscous stress tensor
ϕ	dimensionless	Relative humidity

NOTATION	SI UNIT	DESCRIPTION
ϕ_{amb}	dimensionless	Ambient relative humidity
ϕ	rad	Tilt angle (heat transfer coefficient)
ϕ_{ext}	dimensionless	External relative humidity
ϕ_{d}	dimensionless	Relative humidity, downside
ϕ_{u}	dimensionless	Relative humidity, upside
$\Phi_{\mathrm{d} \to \mathrm{u}}$	kg/s	Mass flow rate, positive direction
$\Phi_{\mathrm{u} \to \mathrm{d}}$	kg/s	Mass flow rate, negative direction
$\varphi(\Omega', \Omega)$	dimensionless	Scattering phase function
φ_{s}	rad	Azimuth angle of the Sun
$\varphi_{\mathrm{d} \to \mathrm{u}}$	$\mathrm{kg/(m^2 \cdot s)}$	Mass flux, positive direction
$\varphi_{\mathrm{u} \to \mathrm{d}}$	$\mathrm{kg/(m^2 \cdot s)}$	Mass flux, negative direction
Ψ	J/kg	Force potential
ω	dimensionless	Specific humidity
Ω	dimensionless	Unit vector of a direction in space
Ω	-	Geometry domain
ω_{b}	1/s	Blood perfusion rate
ω_i	$\mathrm{W/(m^2 \cdot K)}$	Discrete incident radiation vector, i-th component
ξ	$\mathrm{kg/m^3}$	Moisture storage capacity

3

Modeling with the Heat Transfer Module

A variety of modeling techniques are discussed in the following sections:

- Heat Transfer Variables
- Moisture Transport Variables
- Using the Boundary Conditions for the Heat Transfer Interfaces
- Handling Frames in Heat Transfer
- Heat Transfer Consistent and Inconsistent Stabilization Methods
- Heat Transfer and Fluid Flow Coupling
- Boundary Wall Temperature
- Solver Settings
- Plotting Results in Thin Layers Extra Dimensions
- Using Ambient Data

Heat Transfer Variables

In this section:

- Predefined Variables
- Global Variables
- Domain Heat Fluxes
- Out-of-Plane Domain Fluxes
- Boundary Heat Fluxes
- Internal Boundary Heat Fluxes
- Domain Heat Sources
- Boundary Heat Sources
- Line and Point Heat Sources
- Moist Air Variables

Predefined Variables

This section lists some predefined variables that are available to evaluate heat fluxes, sources, and integral quantities used in energy balance. All the variable names begin with the physics interface name (the prefix). By default the Heat Transfer interface prefix is `ht`. As an example, you can access the variable named `tflux` using `ht.tflux` (as long as the physics interface is named `ht`).

TABLE 3-1: HEAT TRANSFER PREDEFINED VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
`dEiInt`	Total Accumulated Heat Rate	Global
`ntfluxInt`	Total Net Heat Rate	Global
`QInt`	Total Heat Source	Global
`WnsInt`	Total Fluid Losses	Global
`dEi0Int`	Total Accumulated Energy Rate	Global
`ntefluxInt`	Total Net Energy Rate	Global
`tflux`	Total Heat Flux	Domains, boundaries
`dflux`	Conductive Heat Flux	Domains, boundaries
`turbflux`	Turbulent Heat Flux	Domains, boundaries
`trlflux`	Translational Heat Flux	Domains, boundaries

TABLE 3-1: HEAT TRANSFER PREDEFINED VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
teflux	Total Energy Flux	Domains, boundaries
not applicable	Radiative Heat Flux	Domains
rflux_u rflux_d rflux_z	Radiative Out-of-Plane Heat Flux	Out-of-plane domains (1D and 2D), boundaries
q0_u q0_d q0_z	Out-of-Plane Inward Heat Flux	Out-of-plane domains (1D and 2D)
ntflux	Normal Total Heat Flux	Boundaries
ndflux	Normal Conductive Heat Flux	Boundaries
ncflux	Normal Convective Heat Flux	Boundaries
ntrlflux	Normal Translational Heat Flux	Boundaries
nteflux	Normal Total Energy Flux	Boundaries
ndflux_u	Internal Normal Conductive Heat Flux, Upside	Interior boundaries
ndflux_d	Internal Normal Conductive Heat Flux, Downside	Interior boundaries
ncflux_u	Internal Normal Convective Heat Flux, Upside	Interior boundaries
ncflux_d	Internal Normal Convective Heat Flux, Downside	Interior boundaries
ntrlflux_u	Internal Normal Translational Heat Flux, Upside	Interior boundaries
ntrlflux_d	Internal Normal Translational Heat Flux, Downside	Interior boundaries
ntflux_u	Internal Normal Total Heat Flux, Upside	Interior boundaries
ntflux_d	Internal Normal Total Heat Flux, Downside	Interior boundaries
nteflux_u	Internal Normal Total Energy Flux, Upside	Interior boundaries
nteflux_d	Internal Normal Total Energy Flux, Downside	Interior boundaries
rflux	Radiative Heat Flux	Boundaries

TABLE 3-1: HEAT TRANSFER PREDEFINED VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
Qtot	Domain Heat Sources	Domains
Qbtot	Boundary Heat Sources	Boundaries
Qltot	Line heat source (Line and Point Heat Sources)	Edges, Points (2D, 2Daxi)
Qptot	Point heat source (Line and Point Heat Sources)	Points
T_dp	Dew Point Temperature	Domains
T_eq	Equivalent Temperature	Domains
psat	Saturation Pressure	Domains
phi	Relative Humidity	Domains
Lv	Latent Heat of Evaporation	Domains

Some of these variables are only available with the Heat Transfer Module (`rflux_u`, `rflux_d`, `rflux_z`, `q0_u`, `q0_d`, and `q0_z`), or when either the CFD Module or the Heat Transfer Module is added (`rflux` and `turbflux`).

Global Variables

This section describes variables defined by integrals. A concise notation denotes the different domains of integration: Ω is the geometry domain, $\partial\Omega_{\mathrm{ext}}$ stands for the exterior boundaries, and $\partial\Omega_{\mathrm{int}}$ for the interior boundaries.

TOTAL ACCUMULATED HEAT RATE

The total accumulated heat rate variable, `dEiInt`, is the variation of internal energy per unit time in the domain:

$$\mathrm{dEiInt} = \frac{d}{dt}\int_{\Omega} \rho E d\omega$$

TOTAL NET HEAT RATE

The total net heat rate, `ntfluxInt`, is the integral of Total Heat Flux over all external boundaries. In the case of a fluid domain, it reads:

$$\text{ntfluxInt} = \int_{\partial\Omega_{\text{ext}}} (\rho\mathbf{u}E - k\nabla T + \mathbf{q}_{\text{r}}) \cdot \mathbf{n}d\sigma$$

This indicates the sum of incoming and outgoing total heat flux through the system.

TOTAL HEAT SOURCE

The total heat source, `QInt`, accounts for all domain sources, interior boundary, edge and point sources, and radiative sources at interior boundaries:

$$\text{QInt} = \int_{\Omega} Q d\omega + \int_{\partial\Omega_{\text{int}}} Q_{\text{b}} d\omega + \int_{\partial\Omega_{\text{int}}} Q_{\text{r}} d\omega$$

TOTAL FLUID LOSSES

The total fluid losses, `WnsInt`, correspond to the work lost by a fluid by degradation of energy. These works are transmitted to the system through pressure work and viscous dissipation:

$$\text{WnsInt} = \int_{\Omega} (\mathbf{u} \cdot \nabla p_{\text{A}}) d\omega + \int_{\Omega} (-\tau{:}\nabla\mathbf{u}) d\omega$$

TOTAL ACCUMULATED ENERGY RATE

The total accumulated energy rate, `dEi0Int`, is the variation of total internal energy per unit time in the domain:

$$\text{dEi0Int} = \frac{d}{dt}\int_{\Omega} \rho E_0 d\omega$$

where the total internal energy, E_0, is defined as

$$E_0 = E + \frac{\mathbf{u} \cdot \mathbf{u}}{2}$$

TOTAL NET ENERGY RATE

The total net energy rate, `ntefluxInt`, is the integral of Total Energy Flux over all external boundaries. In the case of a fluid domain, it reads:

$$\text{ntefluxInt} = \int_{\partial\Omega_{\text{ext}}} (\rho\mathbf{u}H_0 - k\nabla T - \tau\mathbf{u} + \mathbf{q}_{\text{r}}) \cdot \mathbf{n}d\sigma$$

This indicates the sum of incoming and outgoing total energy flux through the system.

HEAT BALANCE

According to Equation 4-137, the following equality between COMSOL Multiphysics variables holds:

```
dEiInt + ntfluxInt = QInt - WnsInt
```

This is the most general form that can be used for time-dependent models. At steady-state the formula is simplified. The accumulated heat rate equals zero, so the total net heat rate (the sum of incoming and outgoing heat rates) should correspond to the heat and work sources:

```
ntfluxInt = QInt - WnsInt
```

The sign convention used in COMSOL Multiphysics for `QInt` is positive when energy is produced (as for a heater) and negative when energy is consumed (as for a cooler). For `WnsInt`, the losses that heat up the system are positive and the gains that cool down the system are negative.

For stationary models with convection by an incompressible flow, the heat balance becomes:

```
ntfluxInt = QInt
```

which corresponds to the conservation of convective and conductive flux as in:

$$\int_{\partial\Omega_{\mathrm{ext}}} \rho \mathbf{u} E \cdot \mathbf{n} d\sigma - \int_{\partial\Omega_{\mathrm{ext}}} k \nabla T \cdot \mathbf{n} d\sigma = Q_{\mathrm{Int}}$$

Depending on the radiation discretization method chosen in **Heat Transfer with Radiation in Participating Media**, the contribution to the heat balance is handled differently. In the definition of `ntfluxInt`, the **Rosseland approximation** defines $\mathbf{q}_\mathrm{r}$, the radiative flux, as an extra contribution to the conductive heat flux. The **P1 approximation** and **Discrete ordinates method**, however, include the radiative source $\nabla \cdot \mathbf{q}_\mathrm{r}$ to Q on the domain, in the variable `QInt`.

ENERGY BALANCE

According to Equation 4-138, the following equality between COMSOL Multiphysics predefined variables holds:

```
dEi0Int + ntefluxInt = QInt
```

In stationary models, `dEi0Int` is zero so the energy balance simplifies into:

```
ntefluxInt = QInt
```

At steady state, and without any additional heat source (`QInt` equal to zero), the integral of the net energy flux on all boundaries of the flow domain, `ntefluxInt`, vanishes. On the other hand, the corresponding integral of the net heat flux does not, in general, vanish. It corresponds instead to the losses from mass and momentum equations, such as `WnsInt` for pressure work and viscous dissipation in fluids. Hence, energy is the conserved quantity, not heat.

Domain Heat Fluxes

On domains the heat fluxes are vector quantities. The definition can vary depending on the active physics nodes and selected properties.

TOTAL HEAT FLUX

On domains the total heat flux, `tflux`, corresponds to the conductive and convective heat flux. For accuracy reasons the radiative heat flux is not included.

See Radiative Heat Flux to evaluate the radiative heat flux.

For solid domains — for example, the solid and biological tissue domains — the total heat flux is defined as:

$$\mathrm{tflux} = \mathrm{trlflux} + \mathrm{dflux}$$

For fluid domains (for example, **Fluid**), the total heat flux is defined as:

$$\mathrm{tflux} = \mathrm{cflux} + \mathrm{dflux}$$

CONDUCTIVE HEAT FLUX

The conductive heat flux variable, `dflux`, is evaluated using the temperature gradient and the effective thermal conductivity:

$$\mathrm{dflux} = -k_{\mathrm{eff}} \nabla T$$

In the general case k_{eff} is the thermal conductivity, k.

For heat transfer in fluids with turbulent flow, $k_{\mathrm{eff}} = k + k_{\mathrm{T}}$, where k_{T} is the turbulent thermal conductivity.

For heat transfer in porous media, k_{eff} is the effective conductivity computed from the solid and fluid conductivities.

For heat transfer in building materials, a latent heat source due to evaporation is included in the conductive heat flux variable:

$$\mathtt{dflux} = -(k_{\mathrm{eff}}\nabla T + L_{\mathrm{v}}\delta_{\mathrm{p}}\nabla(\phi p_{\mathrm{sat}}))$$

TURBULENT HEAT FLUX

The turbulent heat flux variable, `turbflux`, enables access to the part of the conductive heat flux that is due to turbulence.

$$\mathrm{turbflux} = -k_{\mathrm{T}}\nabla T$$

CONVECTIVE HEAT FLUX

The convective heat flux variable, `cflux`, is defined using the internal energy, E:

$$\mathrm{cflux} = \rho\mathbf{u}E$$

The internal energy, E, is defined as:

- $E = C_p T$ for solid domains
- $E = C_p T/\gamma$ for ideal gas fluid domains
- $E = H - p/\rho$ for other fluid domains

where H is the enthalpy defined in Equation 4-5.

TRANSLATIONAL HEAT FLUX

Similar to convective heat flux but defined for solid domains with translation. The variable name is `trlflux`.

TOTAL ENERGY FLUX

The total energy flux, `teflux`, is defined when viscous dissipation is enabled:

$$\mathrm{teflux} = \rho\mathbf{u}H_0 + \mathrm{dflux} + \tau\mathbf{u}$$

where the total enthalpy, H_0, is defined as

$$H_0 = H + \frac{\mathbf{u}\cdot\mathbf{u}}{2}$$

RADIATIVE HEAT FLUX

In participating media, the radiative heat flux, q_r, is not available for analysis on domains because it is more accurate to evaluate the radiative heat source $Q_r = \nabla \cdot q_r$.

Out-of-Plane Domain Fluxes

RADIATIVE OUT-OF-PLANE HEAT FLUX

The radiative out-of-plane heat flux, `rflux`, is generated by the Out-of-Plane Radiation feature.

- In 2D:

$$\text{upside: rflux\_u} = \varepsilon_{\mathrm{u}}\sigma(T^4_{\mathrm{amb,\,u}} - T^4)$$

$$\text{downside: rflux\_d} = \varepsilon_{\mathrm{d}}\sigma(T^4_{\mathrm{amb,\,d}} - T^4)$$

- In 1D:

$$\text{rflux\_z} = \varepsilon_z\sigma(T^4_{\mathrm{amb},\,z} - T^4)$$

OUT-OF-PLANE INWARD HEAT FLUX

The convective out-of-plane heat flux, `q0`, is generated by the Out-of-Plane Heat Flux feature.

- In 2D:

$$\text{upside: q0\_u} = h_{\mathrm{u}}(T_{\mathrm{ext,\,u}} - T)$$

$$\text{downside: q0\_d} = h_{\mathrm{d}}(T_{\mathrm{ext,\,d}} - T)$$

- In 1D:

$$\text{q0\_z} = h_z(T_{\mathrm{ext},\,z} - T)$$

Boundary Heat Fluxes

All the domain heat fluxes (vector quantity) are also available as boundary heat fluxes. The boundary heat fluxes are then equal to the mean value of the heat fluxes on adjacent domains. In addition, normal boundary heat fluxes (scalar quantity) are available on boundaries.

NORMAL TOTAL HEAT FLUX

The variable `ntflux` is defined as:

$$\text{ntflux} = \text{ndflux} + \text{ncflux} + \text{ntrlflux}$$

NORMAL CONDUCTIVE HEAT FLUX

The variable `ndflux` is defined on exterior boundaries as:

- $\text{ndflux} = -\text{dflux\_spatial}(T)$ if the adjacent domain is on the downside,
- $\text{ndflux} = -\text{uflux\_spatial}(T)$ if the adjacent domain is on the upside,

and, on interior boundaries, as:

$$\text{ndflux} = (\text{uflux\_spatial}(T) - \text{dflux\_spatial}(T))/2$$

Frames for the Heat Transfer Equations for a description of spatial and material frames.

NORMAL CONVECTIVE HEAT FLUX

The variable `ncflux` is defined as:

$$\text{ncflux} = \text{mean}(\text{cflux}) \cdot \mathbf{n}$$

NORMAL TRANSLATIONAL HEAT FLUX

The variable `ntrlflux` is defined as

$$\text{ntrlflux} = \text{mean}(\text{trlflux}) \cdot \mathbf{n}$$

NORMAL TOTAL ENERGY FLUX

The variable `nteflux` is defined as:

$$\text{nteflux} = \text{mean}(\text{teflux}) \cdot \mathbf{n} - \text{mean}(\text{dflux}) \cdot \mathbf{n} + \text{ndflux}$$

RADIATIVE HEAT FLUX

On boundaries the radiative heat flux, `rflux`, is a scalar quantity defined as:

$$\text{rflux} = \varepsilon\sigma(T_{\text{amb}}^4 - T^4) + \varepsilon\sigma(G - T^4) + q_{\text{r, net}}$$

where the terms account for surface-to-ambient radiative flux, surface-to-surface radiative flux, and radiation in participating net radiative flux, respectively.

Internal Boundary Heat Fluxes

The internal normal boundary heat fluxes (scalar quantity) are available on interior boundaries. They are calculated using the upside and the downside value of heat fluxes from the adjacent domains.

INTERNAL NORMAL CONDUCTIVE HEAT FLUX, UPSIDE

The variable `ndflux_u` is defined as:

$$\mathrm{ndflux\_u} = \mathrm{uflux\_spatial}(T)$$

INTERNAL NORMAL CONDUCTIVE HEAT FLUX, DOWNSIDE

The variable `ndflux_d` is defined as:

$$\mathrm{ndflux\_d} = \mathrm{dflux\_spatial}(T)$$

Frames for the Heat Transfer Equations for a description of spatial and material frames.

INTERNAL NORMAL CONVECTIVE HEAT FLUX, UPSIDE

The variable `ncflux_u` is defined as:

$$\mathrm{ncflux\_u} = \mathrm{up}(\mathrm{cflux}) \cdot \mathbf{un}$$

INTERNAL NORMAL CONVECTIVE HEAT FLUX, DOWNSIDE

The variable `ncflux_d` is defined as:

$$\mathrm{ncflux\_d} = \mathrm{down}(\mathrm{cflux}) \cdot \mathbf{dn}$$

INTERNAL NORMAL TRANSLATIONAL HEAT FLUX, UPSIDE

The variable `ntrlflux_u` is defined as:

$$\mathrm{ntrlflux\_u} = \mathrm{up}(\mathrm{trlflux}) \cdot \mathbf{un}$$

INTERNAL NORMAL TRANSLATIONAL HEAT FLUX, DOWNSIDE

The variable `ntrlflux_d` is defined as:

$$\mathrm{ntrlflux\_d} = \mathrm{down}(\mathrm{trlflux}) \cdot \mathbf{dn}$$

INTERNAL NORMAL TOTAL HEAT FLUX, UPSIDE

The variable `ntflux_u` is defined as:

$$\mathrm{ntflux\_u} = \mathrm{ndflux\_u} + \mathrm{ncflux\_u} + \mathrm{ntrlflux\_u}$$

INTERNAL NORMAL TOTAL HEAT FLUX, DOWNSIDE

The variable `ntflux_d` is defined as:

$$\mathrm{ntflux\_d} = \mathrm{ndflux\_d} + \mathrm{ncflux\_d} + \mathrm{ntrlflux\_d}$$

INTERNAL NORMAL TOTAL ENERGY FLUX, UPSIDE

The variable `nteflux_u` is defined as:

$$\mathrm{nteflux\_u} = \mathrm{up}(\mathrm{teflux}) \cdot \mathbf{un} - \mathrm{up}(\mathrm{dflux}) \cdot \mathbf{un} + \mathrm{ndflux\_u}$$

INTERNAL NORMAL TOTAL ENERGY FLUX, DOWNSIDE

The variable `nteflux_d` is defined as:

$$\mathrm{nteflux\_d} = \mathrm{down}(\mathrm{teflux}) \cdot \mathbf{dn} - \mathrm{down}(\mathrm{dflux}) \cdot \mathbf{dn} + \mathrm{ndflux\_d}$$

Domain Heat Sources

The sum of the domain heat sources added by different physics features is available in the variable `Qtot`, which is the sum of:

- Q's, which are the heat sources added by the Heat Source (described for the Heat Transfer interface) and Electromagnetic Heat Source (described for the Joule Heating interface in the *COMSOL Multiphysics Reference Manual*) features.
- Q_{met}, which is the metabolic heat source added by the Bioheat feature.
- Q_{r}, which is the radiative heat source added by the Radiation in Participating Media (Heat Transfer Interface) and Radiation in Participating Media (RPM Interface) features.
- Q_{geo}, which is the geothermal heat source added by the Geothermal Heating feature.

In Bioheat, the out-of-plane (heat flux and radiation) and blood contributions are not added to `Qtot` because these are considered to be fluxes.

Boundary Heat Sources

The sum of the boundary heat sources added by different boundary conditions is available in the variable, $Q_{\mathrm{b,tot}}$ (SI unit: $\mathrm{W/m^2}$). This variable `Qbtot` is the sum of:

- Q_{b}, which is the boundary heat source added by the Boundary Heat Source boundary condition.
- Q_{sh}, which is the boundary heat source added by the Boundary Electromagnetic Heat Source boundary condition (described for the Joule Heating interface in the *COMSOL Multiphysics Reference Manual*).
- Q_{s}, which is the boundary heat source added by a Layer Heat Source subfeature of a thin layer, see Layer Heat Source (Thin Layer).

Line and Point Heat Sources

The sum of the line heat sources is available in a variable called `Qltot` (SI unit: W/m).

The sum of the point heat sources is available in a variable called `Qptot` (SI unit: W).

Moist Air Variables

The temperature variable solved by the Heat Transfer interfaces corresponds to the dry bulb temperature. This is the temperature measured by a thermometer with a dry sensor and screening to prevent from deviation due to external radiation like solar radiation.

When the presence of water vapor is accounted for in the model, other temperatures may be considered, depending on vapor pressure.

DEW POINT TEMPERATURE

The dew point temperature of a sample of air with water vapor pressure p_{v} is the temperature to which it must be cooled to become fully saturated.

The variable `T_dp` is defined in Ref. 1 by:

$$p_{\mathrm{v,\,sat}}(\mathtt{T\_dp}) = p_{\mathrm{v}} = \mathtt{phi}\cdot\mathtt{psat}$$

where `phi` is the Relative Humidity variable. See Saturation State for the definition of saturation pressure $p_{\mathrm{v,sat}}$ as a function of temperature. See also Saturation Pressure for the definition of the variable `psat`.

EQUIVALENT TEMPERATURE

The equivalent temperature is obtained by adiabatically condensing all the water vapor of a sample of air with initial vapor pressure p_v. In this process, the latent heat decrease due to total removal of the vapor is balanced by a increase of the sensible heat and temperature.

The variable `T_eq` is approximated in Ref. 1 by:

$$\mathtt{T\_eq} = T + \frac{\mathtt{phi} \cdot \mathtt{psat}}{\gamma_{T\mathrm{eq}}}$$

where `phi` is the Relative Humidity, and $\gamma_{T\mathrm{eq}}$ (SI unit: Pa/K) is the psychrometer constant, defined in Ref. 1 by:

$$\gamma_{T\mathrm{eq}} = \frac{M_\mathrm{a} p C_{p,\,\mathrm{a}}}{M_\mathrm{v} L_\mathrm{v}}$$

where p is the total pressure, $C_{p,\,\mathrm{a}}$ is the heat capacity at constant pressure of dry air at temperature T, L_v is the latent heat of evaporation at temperature T (see Latent Heat of Evaporation), and M_a and M_v are the molar mass of dry air and water vapor, respectively.

See also Saturation Pressure for the definition of the variable `psat`.

These definitions are illustrated on Figure 3-1.

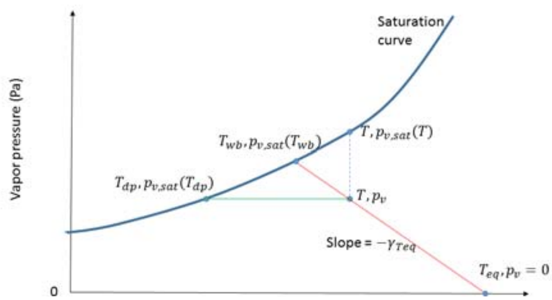


Figure 3-1: Relation between dry bulb, dew point, equivalent, and wet bulb temperatures.

The wet bulb temperature is also represented on Figure 3-1. It is obtained by adiabatically reaching saturation state for a sample of air with initial water vapor pressure p_v. In this process, the latent heat increase due to evaporation is balanced by a decrease of the sensible heat and temperature. It is not available as a predefined variable, but it can be approximated by solving the following equation:

$$T_{\mathrm{wb}} + \frac{p_{\mathrm{v,\,sat}}(T_{\mathrm{wb}})}{\gamma} = T + \frac{\mathtt{phi} \cdot \mathtt{psat}}{\gamma}$$

The psychrometer constant is again evaluated at temperature T.

SATURATION PRESSURE

The variable `psat` is defined by:

$$\mathtt{psat} = \mathtt{fpsat}(T)$$

where T is the temperature. See Functions for the definition of the function `fpsat`.

RELATIVE HUMIDITY

The variable `phi` is defined by:

$$\mathtt{phi} = \frac{p_{\mathrm{v}}}{p_{\mathrm{v,\,sat}}(T)}$$

LATENT HEAT OF EVAPORATION

The variable `Lv` is defined by:

$$\mathtt{Lv} = \mathtt{lv}(T)$$

where T is the temperature. See Functions for the definition of the function `Lv`.

Moisture Transport Variables

Predefined Variables

This section lists the predefined variables that are available with the Moisture Transport interface. All variable names begin with the physics interface name (the prefix). By default, the Moisture Transport interface prefix is `mt`. As an example, you can access the variable named `phi` using `mt.phi` (as long as the physics interface is named `mt`).

TABLE 3-2: MOISTURE TRANSPORT PREDEFINED VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
`phi`	Relative humidity	Domains
`cv`	Vapor concentration	Domains
`cw`	Total moisture concentration	Domains
`cl`	Liquid water concentration	Domains
`psat`	Saturation pressure of water vapor	Domains
`pv`	Partial pressure of water vapor	Domains
`rhol`	Liquid water content	Domains
`rhov`	Vapor content	Domains
`Lv`	Latent heat of evaporation	Domains
`tflux`	Total moisture flux	Domains
`tfluxMag`	Total moisture flux magnitude	Domains
`Gtot`	Total moisture source	Domains
`ntflux`	Normal total moisture flux	Boundaries
`gtot`	Total moisture flux	Boundaries
`ntflux_u`	Normal total moisture flux, upside	Interior boundaries
`ntflux_d`	Normal total moisture flux, downside	Interior boundaries

Moist air properties

PARTIAL AND SATURATION PRESSURES OF WATER VAPOR

The partial pressure of vapor `pv` is the product of the saturation pressure `psat` and the relative humidity `phi`:

$$\mathtt{pv} = \mathtt{phi} \cdot \mathtt{psat}$$

LIQUID, VAPOR, AND MOISTURE CONCENTRATIONS

The total moisture concentration `cw` is the sum of the liquid water and vapor concentrations `cl` and `cv`:

$$\mathtt{cw} = \mathtt{cv} + \mathtt{cl}$$

CONCENTRATIONS AND CONTENTS

The liquid water and vapor contents `rhol` and `rhov` are obtained by multiplying the concentrations `cl` and `cv` by the molar mass of water `Mw`:

$$\mathtt{rhol} = \mathtt{Mw} \cdot \mathtt{cl}$$

$$\mathtt{rhov} = \mathtt{Mw} \cdot \mathtt{cv}$$

LATENT HEAT OF EVAPORATION

The variable `Lv` is defined by:

$$\mathtt{Lv} = \mathtt{lv}(T)$$

where T is the temperature. See Functions for the definition of the function `Lv`.

Domain Moisture Fluxes

TOTAL MOISTURE FLUX

The variable `tflux` is defined as:

$$\mathrm{tflux} = -\xi D_{\mathrm{w}} \nabla\phi - \delta_{\mathrm{p}} \nabla(\phi p_{\mathrm{sat}}(T))$$

TOTAL MOISTURE FLUX MAGNITUDE

The variable `tfluxMag` is defined as:

$$\mathrm{tfluxMag} = \mathrm{norm}(\mathrm{tflux})$$

Boundary Moisture Fluxes

NORMAL TOTAL MOISTURE FLUX

The variable `ntflux` is defined on all boundaries as:

$$\mathrm{ntflux} = \mathrm{mean}(\mathrm{tflux}) \cdot \mathbf{n}$$

INTERNAL NORMAL TOTAL MOISTURE FLUX, UPSIDE

The variable `ntflux_u` is defined on interior boundaries as:

$$\mathrm{ntflux\_u} = \mathrm{up}(\mathrm{tflux}) \cdot \mathbf{un}$$

INTERNAL NORMAL TOTAL MOISTURE FLUX, DOWNSIDE

The variable `ntflux_d` is defined on interior boundaries as:

$$\mathrm{ntflux\_d} = \mathrm{down}(\mathrm{tflux}) \cdot \mathbf{dn}$$

TOTAL MOISTURE FLUX

The sum of the boundary moisture fluxes added by Moisture Flux features is available as the variable `qtot`.

Domain Moisture Source

The sum of the domain moisture sources added by Moisture Source features is available as the variable `Qtot`.

Using the Boundary Conditions for the Heat Transfer Interfaces

In this section:

- Temperature and Heat Flux Boundary Conditions
- Overriding Mechanism for Heat Transfer Boundary Conditions

Temperature and Heat Flux Boundary Conditions

The heat equation accepts two basic types of boundary conditions: specified temperature and specified heat flux. The specified condition is of constraint type and prescribes the temperature on a boundary:

$$T = T_0 \qquad \text{on } \partial\Omega$$

while the latter specifies the inward heat flux

$$-\mathbf{n}\cdot\mathbf{q} = q_0 \qquad \text{on } \partial\Omega$$

where

- $\mathbf{q}$ is the *conductive heat flux vector* (SI unit: $\mathrm{W/m^2}$), $\mathbf{q} = -k\nabla T$.
- $\mathbf{n}$ is the normal vector on the boundary.
- q_0 is the *inward heat flux* (SI unit: $\mathrm{W/m^2}$), normal to the boundary.

The inward heat flux, q_0, is often a sum of contributions from different heat transfer processes (for example, radiation and convection). The special case $q_0 = 0$ is called *thermal insulation*.

A common type of heat flux boundary conditions is one for which $q_0 = h\cdot(T_{\mathrm{ext}} - T)$, where T_{ext} is the temperature far away from the modeled domain and the heat transfer coefficient, h, represents all the physics occurring between the boundary and "far away." It can include almost anything, but the most common situation is that h

represents the effect of an exterior fluid cooling or heating the surface of a solid, a phenomenon often referred to as convective cooling or heating.

The CFD Module and the Heat Transfer Module contain a set of correlations for convective heat flux and heating. See Heat Transfer and Fluid Flow Coupling.

Overriding Mechanism for Heat Transfer Boundary Conditions

Many boundary conditions are available in heat transfer. Some of these can coexist (for example, Heat Flux and Thin Layer); others cannot (for example, Heat Flux and Thermal Insulation).

Several categories of boundary condition exist in heat transfer. Table 3-3 gives the overriding rules for these groups.

1. Temperature, Open Boundary, Open Boundary, Inflow Heat Flux
2. Thermal Insulation, Symmetry, Periodic Condition
3. Heat Flux
4. Boundary Heat Source, Radiation Group, Line Heat Source on Axis
5. Diffuse Mirror, Prescribed Radiosity, Diffuse Surface
6. Opaque Surface, Incident Intensity, Continuity on Interior Boundary
7. Thin Layer, Thermal Contact, Thin Film
8. Isothermal Domain Interface

TABLE 3-3: OVERRIDING RULES FOR HEAT TRANSFER BOUNDARY CONDITIONS

A\B	**1**	**2**	**3**	**4**	**5**	**6**	**7**	**8**
1-Temperature	X	X					X	
2-Thermal Insulation	X	X			X			
3-Heat Flux	X	X						
4-Boundary heat source								
5-Radiation		X			X			
6-Opaque Surface						X		
7-Thin Layer	X						X	
8-Isothermal Domain Interface								X

When there is a boundary condition *A* above a boundary condition *B* in the model tree and both conditions apply to the same boundary, use Table 3-3 to determine if *A* is overridden by *B* or not:

- Locate the line that corresponds to the *A* group (see above the definition of the groups). In the table above only the first member of the group is displayed.
- Locate the column that corresponds to the group of *B*.
- If the corresponding cell is empty, *A* and *B* contribute. If it contains an X, *B* overrides A.

Group 3 and group 4 boundary conditions are always contributing. That means that they never override any other boundary condition. But they might be overridden.

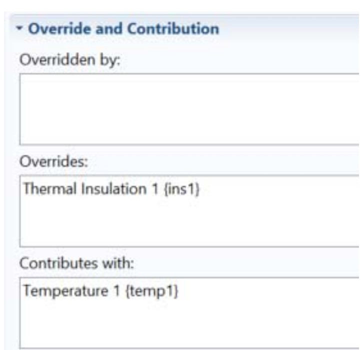
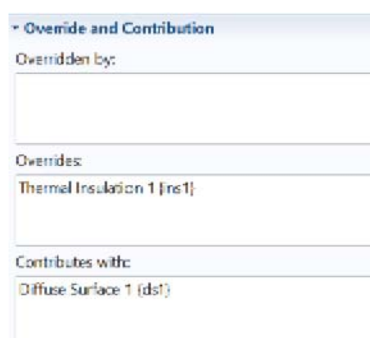
Example 1

Consider a boundary where **Temperature** is applied. Then a **Diffuse Surface** boundary condition is applied on the same boundary afterward.

- **Temperature** belongs to group 1.
- **Diffuse Surface** belongs to group 5.
- The cell on the line of group 1 and the column of group 5 is empty so **Temperature** and **Diffuse Surface** contribute.

This mechanism can be checked on the COMSOL Desktop, in the **Override and Contribution** section of each feature, as shown in the following table:

TABLE 3-4: **Override and Contribution** sections

TEMPERATURE	DIFFUSE SURFACE
	

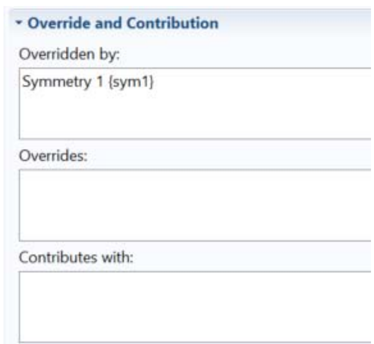
Example 2

Consider a boundary where **Heat Flux** is applied. Then a **Symmetry** boundary condition is applied on the same boundary afterward.

- **Heat Flux** belongs to group 3.
- **Symmetry** belongs to group 2.
- The cell on the line of group 3 and the column of group 2 contains an X so **Heat Flux** is overridden by **Symmetry**.

This mechanism can be checked on the COMSOL Desktop, in the **Override and Contribution** section of each feature, as shown in the following table:

TABLE 3-5: **Override and Contribution** sections

<table>
<tr><th>HEAT FLUX</th><th>SYMMETRY</th></tr>
<tr><td>
</td><td>
</td></tr>
</table>

In Example 2 above, if **Symmetry** followed by **Heat Flux** is added, the boundary conditions contribute.

Handling Frames in Heat Transfer

This section discusses heat transfer analysis with moving frames, when spatial and material frames do not coincide.

- Material and Spatial Frames
- About Frames in the *COMSOL Multiphysics Reference Manual*

When the **Enable conversions between material and spatial frame** check box is selected, all heat transfer interfaces account for deformation effects on heat transfer properties.

The entire physics (equations and variables) are defined on the spatial frame. When a moving mesh is detected, the user inputs for certain features are defined on the material frame and are converted so that all the corresponding variables contain the value on the spatial frame.

Conversion Between Material and Spatial Frames

This subsection contains the list of all heat transfer nodes and the corresponding definition frame:

- Physics Feature Nodes and Definition Frame
- Definition Frame of Domain Nodes
- Definition Frame of Boundary Nodes
- Definition Frame of Edge and Point Nodes

Physics Feature Nodes and Definition Frame

The following explains the different values listed in the *definition frame* column in Table 3-6, Table 3-7, Table 3-8, and Table 3-9:

Material: The inputs are entered by the user and defined on the material frame. Because the heat transfer variables and equations are defined on the spatial frame, the inputs are internally converted to the spatial frame.

Spatial: The inputs are entered by the user and defined on the spatial frame. No conversion is done.

Material/(Spatial): For these physics nodes, select from a menu to decide if the inputs are defined on the material or spatial frame. The default definition frame is the material frame.

(Material)/Spatial: For these physics nodes, select from a menu to decide if the inputs are defined on the material or spatial frame. The default definition frame is the spatial frame.

N/A: There is no definition frame for this physics node.

Definition Frame of Domain Nodes

TABLE 3-6: DOMAIN PHYSICS NODES FOR FRAMES

NODE NAME	**DEFINITION FRAME**
Bioheat	Material
Biological Tissue	Material
Building Material	Spatial
Change Cross Section	Spatial
Change Thickness	Spatial
Fluid	Spatial
Geothermal Heating	Material
Heat Source	Material/(Spatial)
Immobile Fluids	Spatial
Infinite Elements	Spatial
Initial Values	Spatial
Isothermal Domain	Spatial
Opacity	N/A
Out-of-Plane Heat Flux	Spatial
Out-of-Plane Radiation	Spatial
Phase Change Material	Spatial
Porous Medium	Material (Solid part) Spatial (Fluid part)
Pressure Work	Spatial
Radiation in Participating Media	Spatial
Solid	Material
Thermal Dispersion	Spatial
Thermoelastic Damping	Spatial
Translational Motion	Material
Viscous Dissipation	Spatial

Definition Frame of Boundary Nodes

TABLE 3-7: BOUNDARY PHYSICS NODES FOR FRAMES

NODE NAME	DEFINITION FRAME
Boundary Heat Source	Material/(Spatial)
Continuity	Spatial
Continuity on Interior Boundary	Spatial
Diffuse Mirror	Spatial
Diffuse Surface	Spatial
Fracture	Material (Solid part) Spatial (Fluid part)
Heat Flux	(Material)/Spatial
Incident Intensity	Spatial
Inflow Heat Flux	Spatial
Isothermal Domain Interface	Spatial
Layer Heat Source	Material
Opaque Surface	Spatial
Open Boundary	Spatial
Outflow	N/A
Periodic Condition	Spatial
Prescribed Radiosity	Spatial
Radiation Group	N/A
Symmetry	N/A
Temperature	Spatial
Thermal Contact	Material
Thermal Insulation	N/A
Thin Film	Spatial
Thin Layer	Material

The definition frames of the corresponding pair features are identical to the ones of the standard features.

Definition Frame of Edge and Point Nodes

TABLE 3-8: EDGE AND POINT NODES FOR FRAMES

NODE NAME	DEFINITION FRAME
Line Heat Source	Material/(Spatial)
Line Heat Flux	(Material)/Spatial
Point Heat Flux	Spatial
Point Heat Source	Material
Surface-to-Ambient Radiation	Spatial
Temperature	Spatial
Thin Rod	Material

TABLE 3-9: HEAT TRANSFER IN THIN SHELLS NODES

NODE NAME	DEFINITION FRAME
Change Effective Thickness	Spatial
Change Thickness	Spatial
Heat Flux	Spatial/(Material)
Heat Source	Material/(Spatial)
Initial Values	Spatial
Surface-to-Ambient Radiation	Spatial
Temperature	Spatial
Thin Conductive Layer	Material

Heat Transfer Consistent and Inconsistent Stabilization Methods

The different versions of the Heat Transfer interface include the advanced option to set stabilization method parameters. This section has information about these options. To display the stabilization sections, click the **Show** button ( ) and select **Stabilization**.

In this section:

- Consistent Stabilization
- Inconsistent Stabilization

Consistent Stabilization

This section contains two consistent stabilization methods: streamline diffusion and crosswind diffusion. These are consistent stabilization methods, which means that they do not perturb the original transport equation.

The consistent stabilization methods are active by default. A stabilization method is active when the corresponding check box is selected.

Continuous Casting: Application Library path **Heat_Transfer_Module/Thermal_Processing/continuous_casting**

STREAMLINE DIFFUSION

Streamline diffusion is active by default and should remain active for optimal performance for heat transfer in fluids or other applications that include a convective or translational term.

CROSSWIND DIFFUSION

Streamline diffusion introduces artificial diffusion in the streamline direction. This is often enough to obtain a smooth numerical solution provided that the exact solution of the heat equation does not contain any discontinuities. At sharp gradients, however, undershoots and overshoots can occur in the numerical solution. Crosswind diffusion addresses these spurious oscillations by adding diffusion orthogonal to the streamline direction — that is, in the crosswind direction.

Inconsistent Stabilization

This section contains a single stabilization method: isotropic diffusion. Adding isotropic diffusion is equivalent to adding a term to the physical diffusion coefficient. This means that the original problem is not solved, which is why isotropic diffusion is an inconsistent stabilization method. Although the added diffusion definitely attenuates spurious oscillations, try to minimize the use of isotropic diffusion.

By default there is no isotropic diffusion. To add isotropic diffusion, select the **Isotropic diffusion** check box. The field for the tuning parameter δ_{id} then becomes available. The default value is 0.25; increase or decrease the value of δ_{id} to increase or decrease the amount of isotropic diffusion.

In the *COMSOL Multiphysics Reference Manual:*

- Stabilization Techniques
- Stabilization

Heat Transfer and Fluid Flow Coupling

COMSOL Multiphysics offers physics interfaces for heat transfer and fluid flow computations. These interfaces have model inputs that make it possible to couple the physics. In addition, COMSOL Multiphysics and the Heat Transfer Module contain multiphysics coupling interfaces that facilitate the coupling between heat transfer and fluid flow interfaces: **Temperature Coupling**, **Flow Coupling**, and **Non-Isothermal Flow** can be used to couple fluid flow and heat transfer interfaces.

All these options make it possible to build a coupling in different ways. Even if the use of the predefined multiphysics coupling interfaces — **Non-Isothermal Flow** and **Conjugate Heat Transfer** — is the preferred choice, other alternatives can be of interest in particular cases. This section describes the possibility for coupling heat transfer and fluid flow interface and lists the advantages and limitations of each approach.

In this section:

- Coupling Based on Model Inputs
- Temperature Coupling and Flow Coupling Multiphysics Features
- Adding Non-Isothermal Flow Coupling in an Existing Model
- Non-Isothermal Flow and Conjugate Heat Transfer Multiphysics Interfaces

See The Laminar Flow Interface, The Turbulent Flow, Algebraic yPlus Interface, The Turbulent Flow, L-VEL Interface, The Turbulent Flow, k-e Interface, and The Turbulent Flow, Low Re k-e Interface in the *CFD Module User's Guide* for a description of the laminar and turbulent single-phase flow interfaces.

See Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide* for a description of the nodes associated to these interfaces.

Coupling Based on Model Inputs

This option does not use any multiphysics feature. Instead you define the model inputs in each physics interface. You defined the temperature model input in the flow interface and the velocity and pressure model inputs in the heat transfer interface. This approach is valid for laminar flow only. Also, the consistent stabilization does not account for the multiphysics coupling, which can lead to convergence issues when the multiphysics coupling is strong. In addition the physics interface settings may not be optimal for the numerical treatment of the coupling.

One interesting aspect of this method it that the temperature, velocity, or pressure field does not have to be a dependent variable. You can use an analytical expression instead.

Temperature Coupling and Flow Coupling Multiphysics Features

These multiphysics features implement a one-way coupling (one in each direction) between the heat transfer and the fluid flow coupling. These couplings are interesting when you want to explicitly show that the coupling is not bidirectional: it is possible to use only the **Flow Coupling** when the fluid properties are not temperature dependent.

This approach is valid for laminar flow only. Also, the consistent stabilization does not account for the multiphysics coupling, which, similarly to the previous approach, can lead to convergence issues when the multiphysics coupling is strong, in particular when the temperature dependence of the fluid properties is large. In addition, the physics interface settings may not be optimal for the numerical treatment of the coupling.

Adding Non-Isothermal Flow Coupling in an Existing Model

The Heat Transfer Module provides the **Non-Isothermal Flow** multiphysics feature. This feature can be added to a model containing a single-phase flow and a heat transfer interface. It is common to start a model with a single physics (for example, fluid flow), then implement the second one (for example, heat transfer). Then adding the **Non-Isothermal Flow** multiphysics feature realizes the coupling between the two interfaces.

This multiphysics interface handles the two-way coupling. In addition, it accounts for the turbulence in the coupling. In particular, it modifies the effective thermal conductivity and implements thermal wall functions if the fluid flow model requires them. Those modifications affect the implementation of several heat transfer features. This multiphysics feature also redefines the consistent stabilization so that the multiphysics coupling effects are accounted for in the numerical stabilization. Finally some physics features are updated when the **Non-Isothermal Flow** multiphysics feature is active. In particular, the **Interior Fan** and **Screen** fluid-flow features are updated to account for the multiphysics coupling.

Note that the physics interface settings may not be optimal for the numerical treatment of the coupling when the multiphysics feature is added afterward.

Non-Isothermal Flow and Conjugate Heat Transfer Multiphysics Interfaces

These multiphysics interfaces are identical except that they do not have the same default features. Both contain a single-phase flow interface and a heat transfer interface coupled with the **Non-Isothermal Flow** multiphysics feature. So all the benefits of this multiphysics feature (see above) are present when these multiphysics interfaces are used.

In addition, the heat transfer and fluid flow interfaces are set up with optimal interface settings: the discretization order of the heat transfer interface is the same as the one used for the fluid flow interface, and the pseudo time stepping is activated in both interfaces.

For these reasons, the use of these multiphysics interfaces is preferred.

Note that you can do a gradual implementation of the model: It is possible to start from these multiphysics interfaces and to disable the multiphysics feature or one of the physics in a first step and then reactivate them when the first step is validated.

Boundary Wall Temperature

Depending on the model configuration, a single temperature field per boundary may not be sufficient to model accurately the temperature. In some cases, different dependent variables are used to compute the temperature in the wall, at the wall sides, or in the turbulent boundary layer. This section describes when additional degrees of freedom are needed and how they are handled.

The boundary temperature variable called `ht.Tvar` describes the wall temperature. When the wall has a nonconstant temperature across its thickness, this variable contains the average value between the temperatures of the two sides of the walls. The actual definition of `ht.Tvar` depends on the model configuration given in the tables below.

Some features define a local temperature: `ht.feat1.Tvar`. For example, when a boundary heat source is applied on a particular side of the layer, this local variable `ht.bhs1.Tvar` contains either the temperature upside or downside the boundary.

The following list includes existing boundary temperature variables that are available depending on the model configuration:

- `T`: general temperature variable that coincides with the wall temperature in most cases
- `TWall`: wall temperature on a **Wall** node with turbulence
- `TWall_u`: upside wall temperature defined by an **Interior Wall** feature with turbulence
- `TWall_d`: downside wall temperature defined by **Interior Wall** feature with turbulence
- `Tu`: upside temperature of the layer
- `Td`: downside temperature of the layer
- `TExtFace`: external temperature of the layer defined by a **Thin Layer** feature applied on an external boundary with **Resistive** option

The values of these variables depend on the selections where they are defined:

- Case 1: intersection between a **Wall** boundary feature selection and interface exterior boundaries

- Case 2: interface between a fluid domain feature and a solid domain feature where a **Wall** boundary feature is active
- Case 3: interface between a two fluid domain features where an **Interior Wall** boundary feature is active

Depending on the turbulence model selected for the flow, wall functions are used or not:

- No turbulence model: no wall functions
- k-ε and k-ω turbulence models: wall functions
- SST, low Re k-ε, Spalart Allmaras, L-VEL, and Algebraic yPlus turbulence models: no wall functions

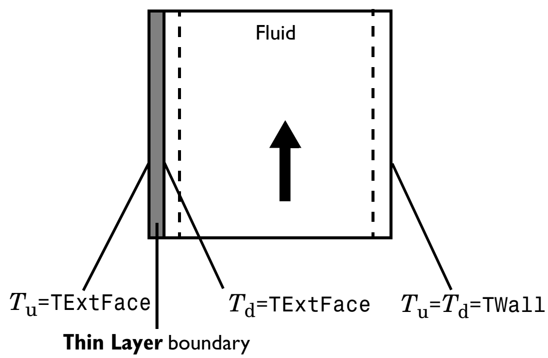
The following sections summarize the definitions of the temperature variables for the different configurations in tables. Some of the configurations listed in the tables are represented in a sketch.

CASE 1

TABLE 3-10: DEFINITIONS OF THE TEMPERATURE VARIABLES FOR CASE 1

<table>
<tr><th rowspan="4"></th><th colspan="6">Case 1: Intersection between a Wall boundary feature selection and exterior boundaries</th></tr>
<tr><th colspan="3">Wall functions</th><th colspan="3">No wall functions</th></tr>
<tr><th rowspan="2">Without Thin Layer</th><th colspan="2">With Thin Layer</th><th rowspan="2">Without Thin Layer</th><th colspan="2">With Thin Layer</th></tr>
<tr><th>Conductive</th><th>Resistive/ General</th><th>Conductive</th><th>Resistive/ General</th></tr>
<tr><td>ht.Tvar</td><td colspan="3">TWall</td><td colspan="3">T</td></tr>
<tr><td>ht.Tu</td><td colspan="2" rowspan="2">TWall</td><td>TExtFace</td><td colspan="2" rowspan="2">T</td><td>TExtFace</td></tr>
<tr><td>ht.Td</td><td>TWall</td><td>T</td></tr>
</table>

The following figure shows a configuration with a fluid domain with a **Thin Layer** feature on the left boundary and a **Wall** feature on the right boundary. This example uses wall functions.



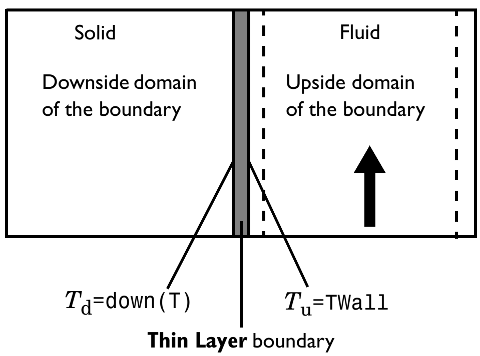
CASE 2 WITH SOLID DOMAIN DOWNSIDE

TABLE 3-11: DEFINITIONS OF THE TEMPERATURE VARIABLES FOR CASE 2 WITH SOLID DOMAIN DOWNSIDE

<table>
<tr><th rowspan="4"></th><th colspan="6">Case 2: Interface between a fluid domain feature and a solid domain feature where a Wall boundary feature is active

The solid is downside</th></tr>
<tr><th colspan="3">Wall functions</th><th colspan="3">No wall functions</th></tr>
<tr><th rowspan="2">Without Thin Layer</th><th colspan="2">With Thin Layer</th><th rowspan="2">Without Thin Layer</th><th colspan="2">With Thin Layer</th></tr>
<tr><th>Conductive</th><th>Resistive/ General</th><th>Conductive</th><th>Resistive/ General</th></tr>
<tr><td>ht.Tvar</td><td colspan="3">down(T)</td><td colspan="3">T</td></tr>
<tr><td>ht.Tu</td><td colspan="2" rowspan="2">down(T)</td><td>TWall</td><td colspan="2" rowspan="2">T</td><td>up(T)</td></tr>
<tr><td>ht.Td</td><td>down(T)</td><td>down(T)</td></tr>
</table>

The following figure shows a configuration with a solid as downside domain and a fluid as upside domain with a **Thin Layer** feature in between. This example uses wall functions.

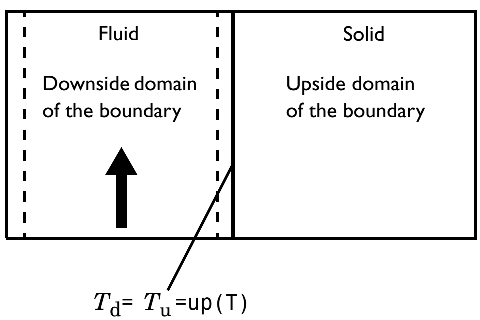


CASE 2 WITH SOLID DOMAIN UPSIDE

TABLE 3-12: DEFINITIONS OF THE TEMPERATURE VARIABLES FOR CASE 2 WITH SOLID DOMAIN UPSIDE

	Case 2: Interface between a fluid domain feature and a solid domain feature where a **Wall** boundary feature is active The solid is upside					
	Wall functions			No wall functions		
	Without Thin Layer	With Thin Layer		Without Thin Layer	With Thin Layer	
		Conductive	Resistive/ General		Conductive	Resistive/ General
`ht.Tvar`	`up(T)`			`T`		
`ht.Tu`	`up(T)`		`up(T)`	`T`		`up(T)`
`ht.Td`			`TWall`			`down(T)`

The following figure shows a configuration with a fluid as downside domain and a solid as upside domain without a **Thin Layer** feature in between. This example uses wall functions.

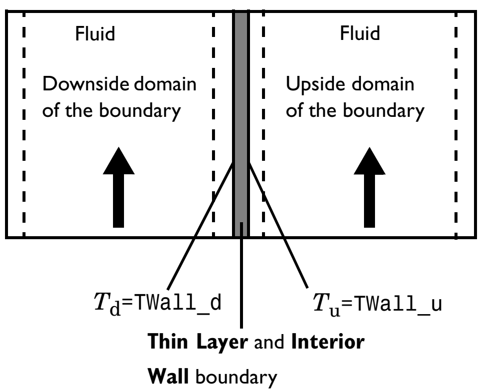


CASE 3

TABLE 3-13: DEFINITIONS OF THE TEMPERATURE VARIABLES FOR CASE 3

	Case 3: Interface between two fluid domain features where an **Interior Wall** boundary feature is active					
	Wall functions			No wall functions		
	Without Thin Layer	With Thin Layer		Without Thin Layer	With Thin Layer	
		Conductive	Resistive/ General		Conductive	Resistive/ General
`ht.Tvar`	`TWall`	`(TWall_u+TWall_d)/2`		`T`		
`ht.Tu`		`TWall_u`		`T`		`up(T)`
`ht.Td`		`TWall_d`				`down(T)`

The following figure shows a configuration with two fluid domains with **Thin Layer** and **Interior Wall** features in between. This example uses wall functions.



Solver Settings

The information about default solvers given below is specific to the Heat Transfer and Moisture Transport interfaces when the **Stationary** and **Time-Dependent** studies are used. A comprehensive description of solver settings and corresponding theory is available in the Study and Study Step Types section of the *COMSOL Multiphysics Reference Manual*.

See also Studies and Solvers in the *COMSOL Multiphysics Reference Manual*

Linearity property of the temperature equation

The Heat Transfer interfaces define an elliptic partial differential equation for the temperature, T, of the form:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = Q$$

$$\mathbf{q} = -k\nabla T$$

with Dirichlet and Neumann boundary conditions at some boundaries:

$$T = T_0$$

$$-\mathbf{n} \cdot \mathbf{q} = q_0$$

In its basic form, the density, ρ, heat capacity, C_p, thermal conductivity, k, heat sources, Q, constraint temperatures, T_0, and heat fluxes, q_0, are all constant, which leads to a linear system. Here, linear solvers described in the next paragraphs are completely suited for the resolution.

However, nonlinearities can appear in the equation in the following cases:

- The material properties, ρ, C_p, and k, have a temperature dependency.
- The heat sources are not linear in temperature.

- The Neumann boundary condition is not linear in temperature, hence
  - A convective cooling condition of type $-\mathbf{n} \cdot \mathbf{q} = h(T_{\mathrm{ext}} - T)$ keeps the linearity of the problem when the heat transfer coefficient, h, is constant.
  - A radiative condition of type $-\mathbf{n} \cdot \mathbf{q} = \varepsilon\sigma({T_{\mathrm{amb}}}^4 - T^4)$ is strongly nonlinear.

Different nonlinear solvers are also provided for these kinds of problems.

Linear Solver

DEFAULT SETTINGS FOR HEAT TRANSFER INTERFACES

The default linear solver is determined based on the number of degrees of freedom and physics interface settings.

For small number of degrees of freedom, the direct PARDISO solver is used. It is known to be robust and fast for small-sized problems.

For larger models, the linear iterative GMRES solver with multigrid preconditioner is used. In most cases, SOR line is the presmoother and postsmoother. This solver is memory effective and fast for large models. When the heat transfer model contains settings that lead to a system matrix with 0 on the diagonal (for example, Lagrange multipliers for weak constraints), SOR line cannot be used and is replaced by Vanka, which is usually slower and uses more memory.

DEFAULT SETTINGS FOR MOISTURE TRANSPORT INTERFACE

The linear iterative GMRES solver with geometric multigrid (GMG) acceleration is used by default. In most cases, SOR line is used as the presmoother and postsmoother for better performance. When weak constraints are set, the use of Lagrange multipliers leads to a system matrix with 0 on the diagonal, and SOR line cannot be used. It is replaced by Vanka, which is usually slower and uses more memory.

TUNING LINEAR SOLVER

Tuning the linear solver may be considered in case of nonconvergence or low performance. When convergence fails you should first verify that this is not due to an ill-posed model, or inappropriate settings in the Time-Dependent study or nonlinear solver.

Several options are available to tune the linear solver settings. This paragraph focuses only on the most commonly used ones.

Switch to PARDISO

When the GMRES solver with multigrid preconditioner is set by default, using PARDISO instead can be considered provided that enough RAM is available. Indeed, PARDISO usually converges easily but uses much more memory than the default iterative solver. If PARDISO does not converge, it may indicate that there is an issue in the model definition or with other solver settings.

Optimize GMRES/Multigrid for Memory

In order to optimize further the memory needed by the iterative solver, the number of mesh elements on the coarser multigrid level can be reduced by, for instance, increasing the **Mesh coarsening factor** or the **Number of multigrid levels**. The latter strategy may also increase the resolution time.

Optimize GMRES/Multigrid for Convergence

When the linear solver has difficulties to converge, the following settings can be tuned:

- When the convergence graph of GMRES shows a slow down every 50 iterations, the **Number of iteration before restart** parameter (default value of 50) should be increased — doubled for example. This may also increase the memory consumption.
- Increasing the **Number of iteration** in the Multigrid settings, and in the presmoother and postsmoother nodes improves the quality of the preconditioner and convergence of GMRES.
- Since an excessive difference between two multigrid levels can affect the convergence, lowering the **Mesh coarsening factor** in the Multigrid settings can help convergence.
- Consider creating the multigrid level meshes manually if the automatic coarsening method fails or leads to poor quality meshes.

Choosing the Right Linear System Solver in the *COMSOL Multiphysics Reference Manual*

Nonlinear Solver

DEFAULT SETTINGS FOR HEAT TRANSFER INTERFACES

Nonlinear solver settings depend on the heat transfer model and on the study type.

Fully Coupled Solver Attribute

Heat transfer models with and without surface-to-surface radiation use a fully coupled nonlinear solver attribute by default. The Jacobian update is set to minimal. A Newton nonlinear method is set by default with

- Automatic damping factor computation for stationary studies
- Constant damping factor for time-dependent studies

Segregated Solver Attribute

The segregated solver attribute is set by default in the following cases:

- Another physics interface is solved together with heat transfer. The dependent variables of the heat transfer interface are placed in a separate segregated group.
- Radiation in participating media using the **Discrete ordinates method** defines a large number of dependent variables (up to 80), which are placed in segregated groups. The number of dependent variables per segregated group and the nonlinear method settings depend on the **Performance index** parameter available in the heat transfer interface settings in the **Participating Media Settings** section.
- The **Biological Tissue** feature with **Include damage integral analysis** option selected defines an additional dependent variable that is placed in a dedicated segregated group. In addition when the **Temperature threshold** option is used, a dependent variable is added to the **Previous solution** step. It uses a direct linear solver. The default nonlinear method is the Newton method with constant damping factor.

DEFAULT SETTINGS FOR MOISTURE TRANSPORT INTERFACE

A Newton nonlinear method is set by default with a constant damping factor (0.8). The **Jacobian update** is set to **On every iteration** for stationary studies and to **Once per time step** for time-dependent studies.

TUNING THE NONLINEAR SOLVER

Default solver settings are defined to handle efficiently classical configurations. For particular applications, the default settings may need modifications to improve the robustness and performance of the solver.

Optimize Nonlinear Solver for Robustness

When the nonlinear solver fails or converges erratically, different options can be considered:

- Using the **Automatic highly nonlinear (Newton)** option forces to start the computation with a very low damping factor and increases it carefully. Alternatively a low constant damping factor can be used. The damping factor ranges between 0 and 1. A constant damping factor equal to 0.1 is a very low value and should be robust but slow to converge. For low values of the damping factor, it is thus usually needed to increase the number of nonlinear iterations. If the nonlinear solver is unstable with such a damping factor then the automatic option should be used because it makes it possible to start with a lower damping factor and gradually increases it.

- A good initial value, as close as possible from the expected solution and consistent with the boundary conditions, helps to guide the nonlinear solver to a stable physical solution. To do that:
  - Try to ramp the temperature on the boundary from the initial to the desired value by using a auxiliary sweep — for stationary problems — or a time-dependent step function — for time-dependent problems.
  - Use results from a simplified problem, for instance with no temperature dependency, or using a one-way multiphysics coupling, as initial value.

  Note that it is sometimes easier to update the boundary conditions than the initial condition to get consistent initial settings (see the Heat Conduction in a Finite Slab model).
- When it is not possible to provide a good initial value, the segregated solver associated with low damping factors in each segregated step helps to achieve convergence.
- Forcing the Jacobian update at every iteration ensures that the nonlinear solver iterates using optimal information from the equation system. This is needed when nonlinearities are due to the temperature itself — for example, in case of strong temperature dependency of material properties — or to another variable solved in the same segregated group as the temperature — for example, in natural convection models.

Optimize Convergence Speed

Low convergence can be improved by following ways:

- Using a constant damping factor equal to 1 for linear problems. The linearity is determined at the beginning of the resolution and indicated in the **Log** section of the solver window.
- Providing a good initial value is an asset for computational speed.

- In the convergence area, the fully coupled solver has a better convergence rate than the segregated solver.
- Using minimal Jacobian update option avoid to spend time in Jacobian computation. This is suited for linear models and models with mild nonlinearities.

Time-Dependent Study

DEFAULT SETTINGS FOR HEAT TRANSFER INTERFACES

The default time-stepping method for the Heat Transfer interfaces is BDF at second order. It excludes algebraic variable from the error estimate.

When the **Biological Tissue** feature is active with the **Include damage integral analysis** option selected, particular settings for the time-dependent solver are used to efficiently compute the damage indicators:

- **BDF** is replaced by **Generalized alpha**.
- The **Absolute tolerance** of the scaled damage indicator dependent variable is set to 1, meaning that these variable are neglected in the error estimate.
- If the **Adaptive mesh refinement** option is selected in the study settings, the error indicator is set to $\sqrt{\nabla\theta_{\mathrm{d,\,sm}} \cdot \nabla\theta_{\mathrm{d,\,sm}}}$ where $\theta_{\mathrm{d,\,sm}}$ is the smoothed indicator of necrotic tissue (the fraction of necrotic tissue, θ_{d}, is discontinuous in general).
- If the **Temperature threshold** option is used in the **Biological Tissue** feature, the instantaneous damage indicator, α_2, is placed in the **Previous solution** step. This setting avoids wrong detection of irreversible damage due to nonlinear iterations that may go through a state where the damage criteria is met and then converge to a solution where the damage criteria is no longer met.

DEFAULT SETTINGS FOR MOISTURE TRANSPORT INTERFACE

The default time-stepping method for the Moisture Transport interface is BDF.

TUNING THE TIME-DEPENDENT SOLVER

The quality of the time-stepping influences the nonlinear solver convergence. Tiny time steps usually lead to mildly nonlinear problems at each time step whereas large time steps can result in (fewer) highly nonlinear problems.

The default solver settings for transient heat transfer defines the maximal number of nonlinear iterations to 5. If this is not sufficient, it is recommended to use smaller time steps and to verify if the model definition does not contain discontinuities in time. If so, consider using smooth step functions to model sharp variations in time.

There are several ways to control the time step size:

- An implicit way is to define a lower relative tolerance in the study settings. When the relative tolerance is lowered, the absolute tolerance should be reduced in the same proportion.
- The most explicit way is to define a maximum time step. This is an appropriate option when the same maximum time step is relevant for the entire simulation. Otherwise, it is possible to include times of interest in the **Times** field of the time-dependent study and to use the **Intermediate** option in the **Time Stepping** settings.
- Lastly you can control the time step by triggering an event when a particular condition is meet (see the documentation about The Events Interface in the *COMSOL Multiphysics Reference Manual*). This advanced method can be efficient when the other simpler methods are not applicable.

It is also recommended to inspect the solver log and check the default scaling of dependent variables in case of convergence failure. In case of incorrect automatic scaling, consider using **Manual** settings in the **Dependent Variable** attribute node.

Time-Dependent Solver in the *COMSOL Multiphysics Reference Manual*

Guidelines for Solving Surface-to-Surface Radiation Problems

The following guidelines are helpful when selecting solver settings for models that involve surface-to-surface radiation:

- Surface-to-surface radiation makes the Jacobian matrix of the discrete model partly filled as opposed to the usual sparse matrix. The additional nonzero elements in the matrix appear in the rows and columns corresponding to the radiosity degrees of freedom. It is therefore common practice to keep the element order of the radiosity variable, J, low. By default, linear Lagrange elements are used irrespective of the shape-function order specified for the temperature. When you need to increase the resolution of your temperature field, it might be worth considering raising the order of the temperature elements instead of refining the mesh.
- The **Assembly block size** parameter (found in the **Advanced** solver feature) can have a major influence on memory usage during the assembly of problems where surface-to-surface radiation is enabled. When surface-to-surface is detected, the

solver sets the assembly block size at 100. Using a smaller block size also leads to more frequent updates of the progress bar.

Introduction to Solvers and Studies and Advanced in the *COMSOL Multiphysics Reference Manual*

Multiphysics

MULTIPHYSICS MODELS

Unless the model contains a multiphysics node that defines a coupling between a Heat Transfer interface and another interface (see Multiphysics Couplings below), each physics interface defines default solver settings that are merged.

The Heat Transfer interfaces always define a dedicated segregated group that uses a linear solver optimized for the heat transfer equations. For strongly coupled models, it may be efficient to merge two (or more) segregated steps. In this case, a unique linear solver must be chosen for the fully coupled solver or the new segregated group.

Time-dependent settings from different physics interfaces may compete. When the different settings are merged the strictest one is kept.

MULTIPHYSICS COUPLINGS

When a Heat Transfer interface is coupled with another physics interface through a multiphysics coupling feature, additional predefined default settings are loaded. The next two paragraphs describes the subtleties of the Non-Isothermal Flow, Electromagnetic Heating, and Heat and Moisture Transport interfaces.

Non-Isothermal Flow

The Non-Isothermal Flow multiphysics coupling controls the solver settings for the flow and the temperature-dependent variables.

When it assumes a weak coupling between the flow and the heat interfaces (typically no Volume Force feature in the flow interface), the default solver contains dedicated segregated groups for heat and flow dependent variables. Each uses the default linear solver of the corresponding interface.

When a strong coupling is assumed (at least one Volume Force feature in the flow interface), the default solver merges the temperature, pressure, and velocity. In this case, the linear solver corresponds to the default linear solver of the flow interface. The Jacobian is updated once per time step.

Electromagnetic Heating

The Electromagnetic Heating multiphysics interfaces (Joule heating, Laser Heating, Induction Heating, and Microwave Heating) define default settings that solve the temperature and the electromagnetic fields using a coupled step. It can be the fully coupled nonlinear solver if there is no additional variable to solve for, otherwise it is a segregated step containing the temperature and the electromagnetic variables. However when radiation in participating media or damage variable are solved they are placed in a separate group as described above.

Heat and Moisture Transport

The Heat and Moisture Transport interface defines dedicated segregated groups for heat and moisture dependent variables. Each uses the default linear solver of the corresponding interface. A **Lower Limit** node is added for the relative humidity to enforce a positive value.

Plotting Results in Thin Layers Extra Dimensions

When modeling thin layers with extra dimensions — if the **Layer type** is set to **General** in the **Thin Layer** feature of the Heat Transfer interface, or when using the **Thin Layered Shell** feature of the Heat Transfer in Thin Shells interface — a new shape function, `Txdim`, is automatically created. This variable corresponds to the temperature field in the layer. Because the layer is not explicitly represented in the geometry, dedicated tools are available. There are two ways of plotting the temperature in the 1D extra dimension:

- Along the layer, for a fixed coordinate between 0 and d_s (the layer thickness) in the 1D extra dimension.
- Through the thin layer, at one position on the boundary

Theory for Heat Transfer in Thin Structures

Thin Layer

Thin Layered Shell (Heat Transfer in Thin Shells Interface)

Using Extra Dimensions in the *COMSOL Multiphysics Reference Manual*.

Along the Layer

The extra dimension temperature `Txdim` can be evaluated along the layer by using the `atxd1` operator, with the expression `xdimTag.atxd1(xd, expr)`, where:

- `xdimTag` is the extra dimension tag. For example, it can be `ht_tl1_xdim5_xdim`, in the case of a thin layer (`tl1`) with five layers (`xdim5`) in a physics interface with tag `ht`.
- `xd` is the coordinate in the extra dimension. It varies from 0 to d_s, which is the sum of layer thicknesses. By convention, `xd=0` corresponds to the upside of the boundary where the thin layer is defined, whereas `xd=`d_s corresponds to its downside. Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. See Tangent

and Normal Variables in the *COMSOL Multiphysics Reference Manual*. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.

- `expr` is the quantity to be evaluated at the point `xd`. For example, it can be set to `ht.tl1.Txdim` to evaluate the temperature. There are others post-processing variables defined on the extra dimension that can be found in the **Equation View** subnode of **Thin Layer.**

In 2D and 2D axisymmetric geometries, the section is represented in a line graph under a 1D plot group, whereas it is a surface plot for 3D geometries.

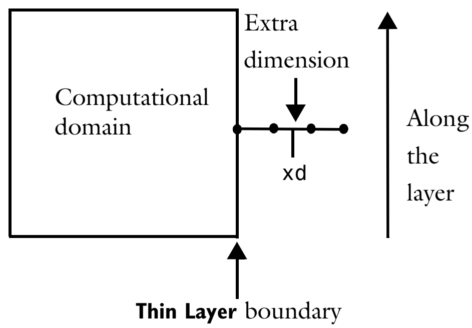


Figure 3-2: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results along the layer at the coordinate `xd`.

Through the Thin Layer

The extra dimension temperature `Txdim` can be evaluated through the thin layer by using the operators `atxd1` and `atxd2`.

In 2D and 2D axisymmetric geometries, use `compTag.atxd1(x0, y0, expr)`, where:

- `compTag` is the component tag. In most cases, this tag is `comp1`. It is possible to check it in the **Properties** window of the component node (display it by right-clicking on the node and selecting **Properties**).
- `x0` and `y0` are the coordinates of the point in the base geometry that belongs to the boundary linked with the extra dimension. Note that these are 2D coordinates from the global coordinate system and not curvilinear coordinates.
- `expr` is the quantity to be evaluated at the point (`x0`, `y0`). For example, it can be set to `ht.tl1.Txdim` to evaluate the temperature.

In 3D geometries, use `compTag.atxd2(x0, y0, z0, expr)`, where:

- `compTag` is the component tag. For example, it can be `comp1`.
- `x0`, `y0`, and `z0` are the coordinates of the point in the base geometry that belongs to the boundary linked with the extra dimension.
- `expr` is the quantity to be evaluated at the point (`x0`, `y0`, `z0`). For example, it can be set to `ht.tl1.Txdim` to plot the temperature. Note that here too, these are 3D coordinates from the global coordinate system and not curvilinear coordinates.

For all dimensions, the section is represented in a line graph under a 1D plot group. In order to use this, the data set solution has to select the extra dimension as component.

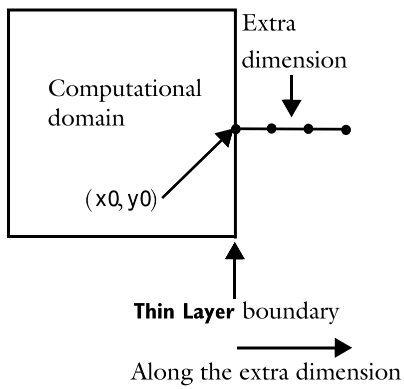


Figure 3-3: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results through the layer at the point (`x0,y0`*).*

Using Ambient Data

The ambient data available within the Heat Transfer interfaces come from the processing of measured data from ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org), given as frequencies of observations, monthly and hourly averaged for several past years of observation. They provide time-dependent weather conditions for more than 6000 stations worldwide in terms of the dry bulb temperature, the dew point temperature, the relative humidity, the wind speed, and the solar irradiance.

See Ambient Settings for the settings related to the ambient variables in the Heat Transfer (`ht`) and Heat Transfer in Thin Shells (`htsh`) interfaces.

> When no special mention is added, the term temperature stands for the dry bulb temperature. See Moist Air Variables for the definition of the dry bulb temperature, the dew point temperature, and the relative humidity.

Processing of ASHRAE data

From frequencies of measured values, a weighted mean of the data and a standard deviation from the weighted mean are computed for each month. For the temperature, more data are available and the weighted mean and standard deviation are also computed at each hour. Figure 3-4 shows an example of weighted mean computation for the diurnal temperature fluctuations from the initial data given as frequencies of observations.

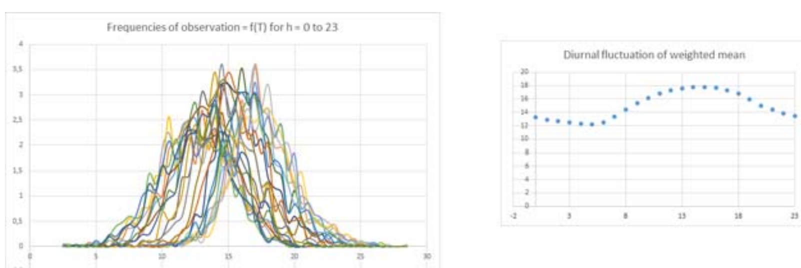


Figure 3-4: Computation of weighted mean from frequencies of observations for the diurnal fluctuations of temperature.

These values are used for the definition of different conditions, as detailed in Ambient Variables and Conditions.

All the monthly averaged observations except the solar irradiance are supposed to be made at the middle of each month. This time depends on the number of days in the month:

- Months with 31 days (January, March, May, July, August, October, December): data at the 16th at noon
- Months with 30 days (April, June, September, November): data at the 16th at midnight
- Months with 29 days (February, leap years): data at the 15th at noon
- Months with 28 days (February, other years): data at the 15th at midnight

In addition, the temperature observations are supposed to be made at the beginning of each hour (00:00 a.m. to 11:00 p.m.).

Finally, the solar irradiance observations are made at the 21st of each month at noon. Depending on the number of days in the month, this date corresponds to 68% (for months with 31 days), 70% (for months with 30 days), or 75% (for February) of the month. The leap years are not considered and the 21st of February always corresponds to 75% of this month.

Ambient data interpolation

The temperature is the only variable for which hourly data are available in addition to monthly averages. So for the temperature a double interpolation is performed to get a temperature profile for every time in a year depending on the date and the hour. For other data the interpolation is based on date only. This provides:

- the annual fluctuation of the dew point temperature, the relative humidity, the wind speed, and the solar irradiance
- the annual and diurnal fluctuation of the temperature.

In all cases the interpolation is of second order, with continuous first order derivative.

Ambient Variables and Conditions

The observed values are processed by computing weighted means, standard deviations, maximum, and minimum, to define different conditions for the temperature, dew point temperature, and wind speed.

CONDITIONS OF TEMPERATURE

- Average:

$$T_{\text{amb}} = \langle T_{\text{station}} \rangle$$

- Low:

$$T_{\text{amb}} = \langle T_{\text{station}} \rangle - \sigma_{T,\,\text{station}}$$

- High:

$$T_{\text{amb}} = \langle T_{\text{station}} \rangle + \sigma_{T,\,\text{station}}$$

- Lowest:

$$T_{\text{amb}} = \min(T_{\text{station}})$$

- Highest:

$$T_{\text{amb}} = \max(T_{\text{station}})$$

- User defined coefficient for deviation:

$$T_{\text{amb}} = \langle T_{\text{station}} \rangle + c_{\sigma} \cdot \sigma_{T,\,\text{station}}$$

- User defined correction:

$$T_{\text{amb}} = \langle T_{\text{station}} \rangle + \Delta T$$

where:

- $<T_{\text{station}}>$ (SI unit: K) is the weighted mean of the observed values of temperature at the station.
- $\sigma_{T,\text{station}}$ (SI unit: K) is the standard deviation of the observed values of temperature at the station.
- T_{station} (SI unit: K) is the set of the observed values of temperature at the station.
- c_{σ} (dimensionless) is a user defined multiplicative coefficient applied to $\sigma_{T,\text{station}}$.
- ΔT (SI unit: K) is a user defined additive correction applied to $<T_{\text{station}}>$.

All these conditions are illustrated on Figure 3-5 for the variation of temperature over 1 day at New York/John F. Ke, on the 1st of June.

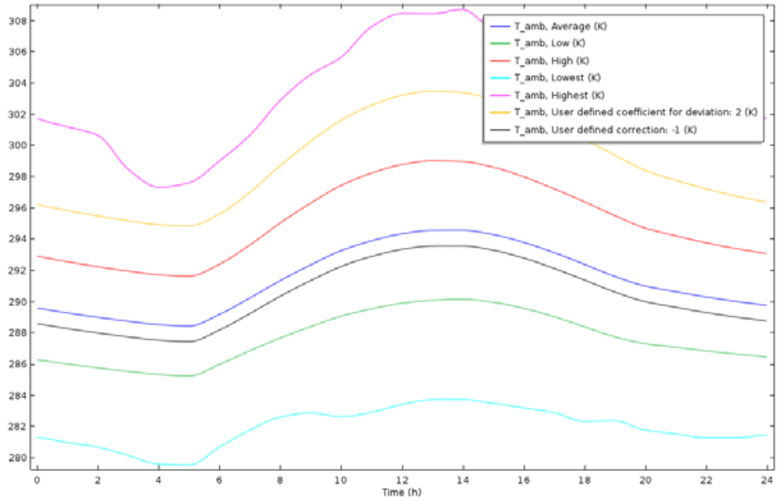


Figure 3-5: Comparison of ambient conditions for the temperature at New York/John F. Ke, on the 1st of June

Additional conditions are defined from observed couples of temperature and wind speed and direction values:

- Heating wind correlation:

$$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle + \Delta T_{\mathrm{wind}}$$

- Cooling wind correlation:

$$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle - \Delta T_{\mathrm{wind}}$$

where ΔT_{wind} (SI unit: K) is an additive correction applied to $<T_{\mathrm{station}}>$, defined as

$$\Delta T_{\mathrm{wind}} = \frac{1}{2}\max(\Delta T_{\mathrm{ws,\,station}}, \Delta T_{\mathrm{wd,\,station}})$$

where $\Delta T_{\mathrm{ws,station}}$ (SI unit: K) and $\Delta T_{\mathrm{wd,station}}$ (SI unit: K) are respectively the maximal variations of observed values of temperature correlated with a set of wind speed and direction observed values.

The heating and cooling wind correlations are illustrated on Figure 3-6 for the variation of temperature over 1 day, at New York/John F. Ke, on the 1st of June.

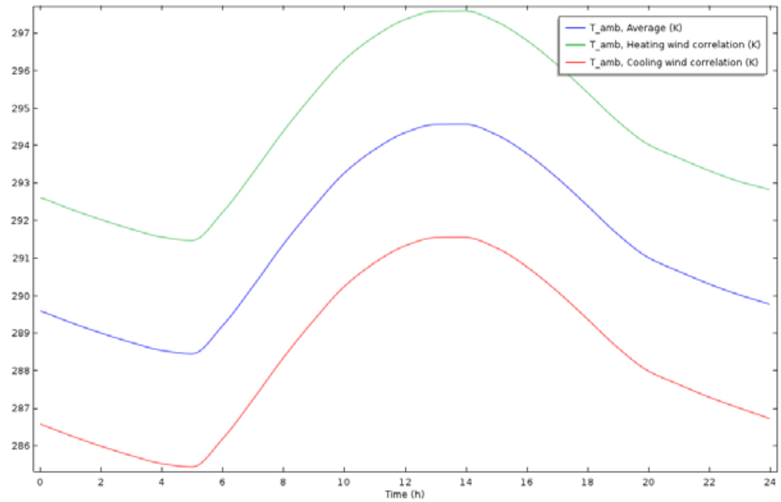


Figure 3-6: Comparison of heating and cooling wind correlations for the temperature at New York/John F. Ke, on the 1st of June

CONDITIONS OF DEW POINT TEMPERATURE

- Average:

$$DPT_{\mathrm{amb}} = \langle DPT_{\mathrm{station}} \rangle$$

- Low:

$$DPT_{\mathrm{amb}} = \langle DPT_{\mathrm{station}} \rangle - \sigma_{DPT,\,\mathrm{station}}$$

- High:

$$DPT_{\mathrm{amb}} = \langle DPT_{\mathrm{station}} \rangle + \sigma_{DPT,\,\mathrm{station}}$$

- Lowest:

$$DPT_{\mathrm{amb}} = \min(DPT_{\mathrm{station}})$$

- Highest:

$$DPT_{\mathrm{amb}} = \max(DPT_{\mathrm{station}})$$

where:

- $\langle DPT_{\text{station}} \rangle$ (SI unit: K) is the weighted mean of the observed values of dew point temperature at the station.
- $\sigma_{DPT,\text{station}}$ (SI unit: K) is the standard deviation of the observed values of dew point temperature at the station.
- DPT_{station} (SI unit: K) is the set of the observed values of dew point temperature at the station.

All these conditions are illustrated on Figure 3-7 for the variation of the dew point temperature over 1 year at New York/John F. Ke.

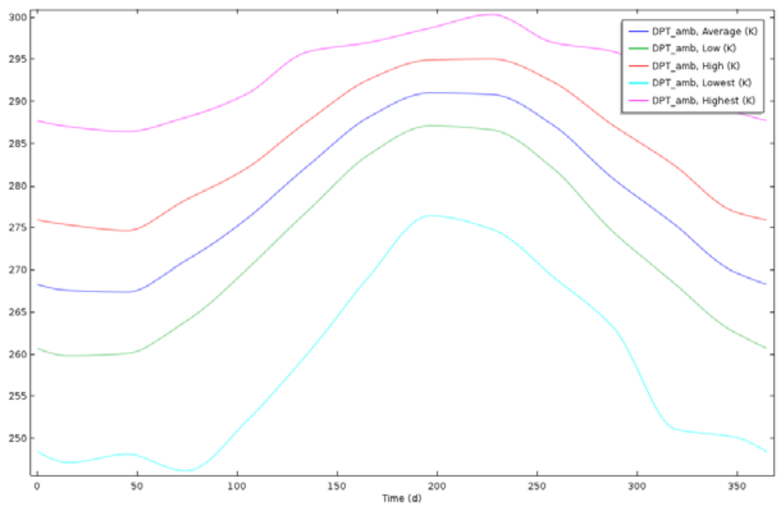


Figure 3-7: Comparison of the ambient conditions for the dew point temperature at New York/John F. Ke

CONDITIONS OF WIND SPEED

- Average:

$$v_{\text{amb}} = \langle v_{\text{station}} \rangle$$

- Low:

$$v_{\text{amb}} = \langle v_{\text{station}} \rangle - \sigma_{v,\,\text{station}}$$

- High:

$$v_{\text{amb}} = \langle v_{\text{station}} \rangle + \sigma_{v,\,\text{station}}$$

- Lowest:

$$v_{\text{amb}} = \min(v_{\text{station}})$$

- Highest:

$$v_{\text{amb}} = \max(v_{\text{station}})$$

where:

- $<v_{\text{station}}>$ (SI unit: m/s) is the weighted mean of the observed values of wind velocity at the station.
- $\sigma_{v,\text{station}}$ (SI unit: m/s) is the standard deviation of the observed values of wind velocity at the station.
- v_{station} (SI unit: m/s) is the set of the observed values of wind velocity at the station.

All these conditions are illustrated on Figure 3-8 for the variation of the wind speed over 1 year at New York/John F. Ke.

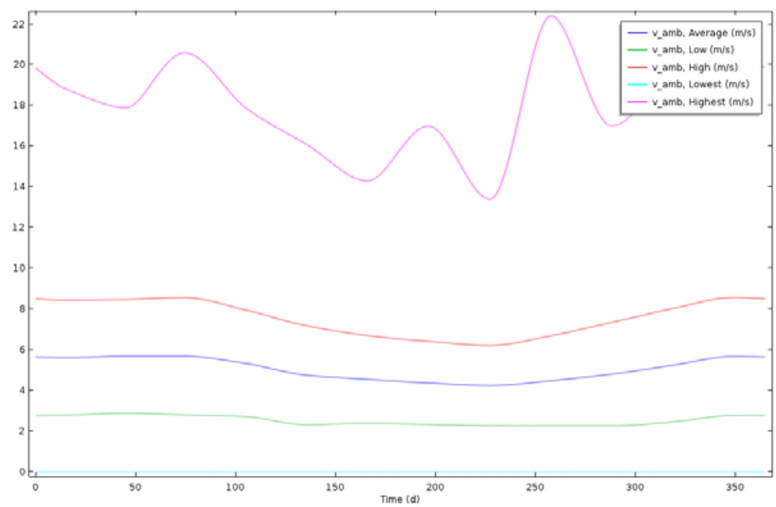


Figure 3-8: Comparison of the ambient conditions for the wind speed at New York/John F. Ke

PRESSURE

$$p_{\text{amb}} = p_{\text{station}}$$

where p_{station} (SI unit: Pa) is the observed value of absolute pressure at the station. Only a single value is available so this data does not vary with time.

RELATIVE HUMIDITY

The relative humidity ϕ_{amb} (dimensionless) is computed from the temperature T_{amb} and the dew point temperature DPT_{amb} with the following relation:

$$\phi_{amb} = \frac{p_{v,\,sat}(DPT_{amb})}{p_{v,\,sat}(T_{amb})}$$

where $p_{v,sat}(T_{amb})$ is the saturation pressure of vapor at T_{amb}.

See Relative Humidity for more details.

As the diurnal variation of temperature is available, the diurnal fluctuations of relative humidity can be computed, as illustrated on Figure 3-9 for New York/John F. Ke, on the 1st of June, for different ambient conditions.

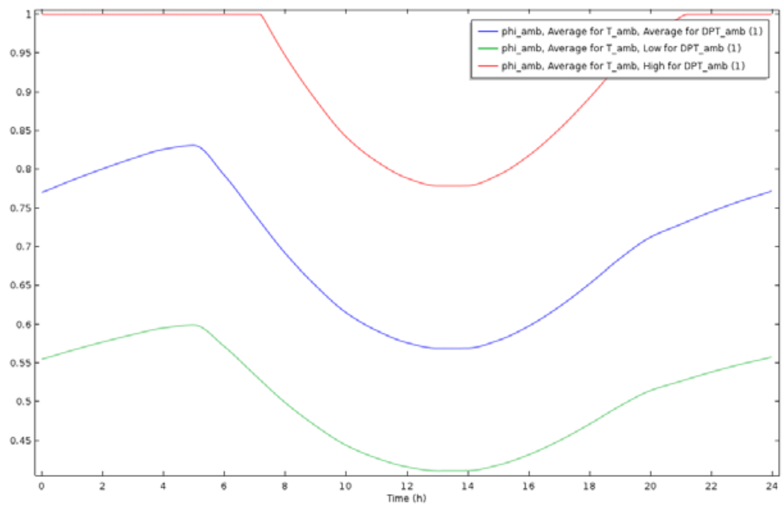


Figure 3-9: Diurnal fluctuations of relative humidity for different ambient conditions at New York/John F. Ke, on the 1st of June

Several conditions of relative humidity can be obtained from the selection of conditions for the temperature and dew point temperature. For consistency, the relative humidity is majored by 1, as shown on the red curve of Figure 3-9.

MOISTURE CONTENT

The moisture content $x_{vap,amb}$ (dimensionless) is computed from the temperature T_{amb}, the absolute pressure p_{amb}, and the relative humidity ϕ_{amb} with the following relation:

$$x_{\text{vap, amb}} = \frac{\phi_{\text{amb}} \cdot p_{v,\text{ sat}}(T_{\text{amb}})}{p_{\text{amb}} - \phi_{\text{amb}} \cdot p_{v,\text{ sat}}(T_{\text{amb}})} \cdot \frac{M_{\text{v}}}{M_{\text{a}}}$$

where $p_{\text{v,sat}}(T_{\text{amb}})$ is the saturation pressure of vapor at T_{amb}, and M_{v} and M_{a} are the molar masses of water vapor and dry air.

See Moisture Content for more details.

SOLAR IRRADIANCE

With clear sky conditions, the noon solar irradiance is essentially provided by the beam normal irradiance, coming directly from the sun. However, the diffuse horizontal irradiance may be also considered.

- If the horizontal diffuse solar irradiance is included:

$$I_{\text{s, amb}} = I_{\text{sn, station}} + I_{\text{sh, station}}$$

- Else:

$$I_{\text{s, amb}} = I_{\text{sn, station}}$$

where $I_{\text{sn,station}}$ (SI unit: W/m^3) and $I_{\text{sh,station}}$ (SI unit: W/m^3) are respectively the observed values of the clear sky noon beam normal and horizontal diffuse solar irradiance.

Figure 3-10 illustrates the evolution of solar irradiance for New York/John F. Ke, over the year.

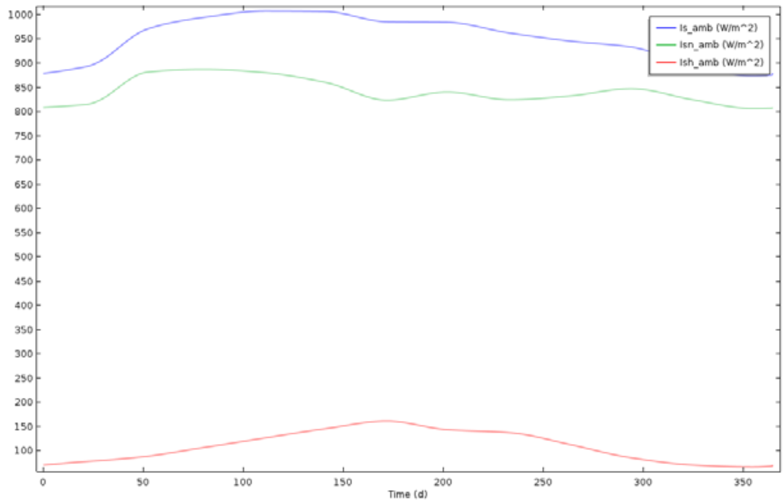


Figure 3-10: Decomposition of solar irradiance into normal and horizontal irradiance at New York/John F. Ke

Use of ambient data from the features

The ambient variables defined in the physics interface are available as inputs for several boundary and initial conditions. For example, when defining the heat transfer coefficient for a **Heat Flux** boundary condition by using a correlation for external forced convection, the wind velocity v_{amb}, the ambient absolute pressure p_{amb}, and the ambient temperature T_{amb} can be selected as inputs:

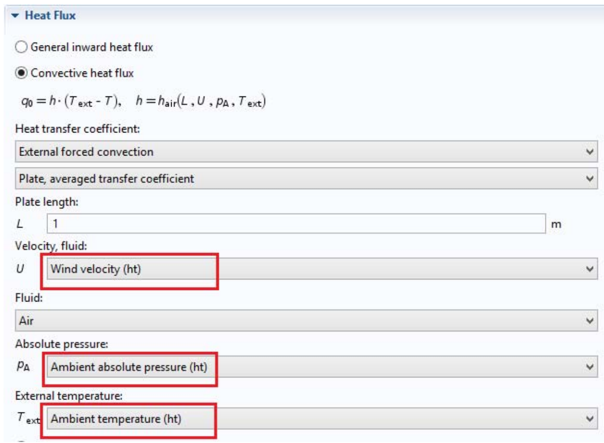


Figure 3-11: Use of ambient data from Heat Flux feature

For an example of the use of user-defined ambient data, see:

Temperature Field in a Cooling Flange: Application Library path **Heat_Transfer_Module/Thermal_Processing/cooling_flange**

For an example of the use of meteorological ambient data, see:

Condensation Detection in an Electronic Device with Transport and Diffusion: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/condensation_electronic_device_transport_diffusion**

Condensation Detection in an Electronic Device: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/condensation_electronic_device**

References

1. J.L. Monteith and M.H. Unsworth, *Principles of Environmental Physics,* Edward Arnold, London, 290 pp., 1990.

4

Theory for the Heat Transfer Module

This chapter details the theory of the physics interfaces, multiphysics couplings, and features found under the **Heat Transfer** branch ( ).

In this chapter:

- Theory for Thermal Contact
- Moist Air Fluid Type
- Temperature Dependence of Surface Tension
- Out-of-Plane Heat Transfer
- The Heat Transfer Coefficients
- Equivalent Thermal Conductivity Correlations
- Heat Flux and Heat Balance
- Frames for the Heat Transfer Equations
- References

Foundations of the General Heat Transfer Equation

This section presents basic results leading to the heat transfer equations solved in COMSOL Multiphysics. Starting by a presentation of heat as another mode of energy transfer different from work, the energy conservation laws that apply lead to the establishment of The Heat Balance Equation. The latter is further derived to give the different versions of the heat transfer equations in various media.

In this section:

Thermodynamic Description of Heat Transfer

In continuum mechanics, a domain Ω is submitted to variations of its kinetic energy due to some external forces according to an equation of motion. The study of such phenomena is covered by solid mechanics and fluid mechanics and the theories behind can be found in the *Structural Mechanics Module User's Guide* and *CFD Module User's Guide*. From an energy point of view, the aforementioned description is incomplete because it does not include heat as another form of energy transfer due to microscopic vibration and interactions of particles. The laws of thermodynamics introduce several concepts to define heat transfer consistently with mechanical energy. In the next paragraphs, a concise presentation of the theory adapted to the use of COMSOL Multiphysics is given. More materials and details are provided in the references listed in the References section.

EXTENSIVE PARAMETERS CHARACTERIZING A SYSTEM

A homogeneous fluid taking place in a domain Ω is characterized by the knowledge of three extensive parameters:

- The *entropy*, S_Ω (SI unit: J),

- The *volume*, V_Ω (SI unit: m^3),
- The *mass*, M_Ω (SI unit: kg).

The *internal energy*, E_Ω (SI unit: J), is an extensive state function of these three variables. It measures the amount of energy in the system excluding kinetic energy and potential energy from external applied forces and is the subject of conservation laws more detailed in The Heat Balance Equation section. To fit with the finite element method solved by COMSOL Multiphysics, specific quantities per unit mass are preferred:

$$S = \frac{S_\Omega}{M_\Omega} \qquad \nu = \frac{V_\Omega}{M_\Omega}$$

The specific internal energy, E (SI unit: J/kg), is then a function of specific entropy, S, and specific volume, ν, related to E_Ω by:

$$E(S, \nu) = \frac{1}{M_\Omega} E_\Omega(S_\Omega, V_\Omega, M_\Omega)$$

For a solid, the specific internal energy, $E(S, F)$, is a function of entropy and deformation gradient, F.

Internal energy is related to the *enthalpy*, H, via the following for a fluid:

$$H = E + \frac{p}{\rho}$$

or the following for a solid (7.33 in Ref. 1):

$$H = E - \mathbf{P}:\mathbf{F}$$

Compared to the internal energy, the enthalpy also includes the pressure-volume potential energy, p/ρ, necessary for instance in volume expansion after an isobaric transformation.

FIRST-ORDER PARAMETERS

The variations of internal energy correspond to variations of entropy and volume according to:

$$dE = \left(\frac{\partial E}{\partial S}\right)_\nu dS + \left(\frac{\partial E}{\partial \nu}\right)_S d\nu$$

First-order parameters are partial derivatives of the specific internal energy. They correspond to the thermodynamic definitions of temperature and pressure:

$$T = \left(\frac{\partial E}{\partial S}\right)_{\mathrm{v}} \qquad p = -\left(\frac{\partial E}{\partial \mathrm{v}}\right)_S \tag{4-1}$$

These lead to the fundamental thermodynamic relation:

$$dE = TdS - pd\mathrm{v}$$

> Temperature is the measurable quantity that gives a phenomenological description of heat transfer. When expressed in kelvin (K), the Second Law of Thermodynamics ensures that T can only take positive values.

Similar relations as those of Equation 4-1 hold for solids:

$$T = \left(\frac{\partial E}{\partial S}\right)_F \qquad P = \left(\frac{\partial E}{\partial F}\right)_S \tag{4-2}$$

$$dE = TdS + P{:}dF$$

Here, the counterpart of the fluid pressure is the first Piola-Kirchhoff stress tensor, P.

SECOND ORDER PARAMETERS

Second order parameters correspond to second partial derivatives of the specific internal energy and provide a various number of thermodynamic coefficients. These are usually given as material properties of the domain material. Among them, the heat capacity at constant pressure and the coefficient of thermal expansion are most often provided. For a fluid, these are

$$C_p = \frac{T}{\left(\frac{\partial T}{\partial S}\right)_{\mathrm{v}}} \qquad \alpha_p = \frac{1}{\mathrm{v}\left(\frac{\partial T}{\partial \mathrm{v}}\right)_S} \tag{4-3}$$

and for a solid, the definitions become:

$$C_p = \frac{T}{\left(\frac{\partial T}{\partial S}\right)_F} \qquad \alpha = F^{-1}\left(\frac{\partial T}{\partial F}\right)_S^{-1} \tag{4-4}$$

Specific heat capacity at constant pressure is the amount of energy required to raise one unit of mass of a substance by one degree while maintained at constant pressure. This quantity is also commonly referred to as *specific heat* or *specific heat capacity*.

The heat capacity at constant pressure and coefficient of thermal expansion are related to the enthalpy, seen as a function of T and p (or P), according to:

$$\left(\frac{\partial H}{\partial T}\right)_p = C_p \qquad \left(\frac{\partial H}{\partial p}\right)_T = \nu(1-\alpha_p T)$$

$$\left(\frac{\partial H}{\partial T}\right)_P = C_p \qquad \left(\frac{\partial H}{\partial P}\right)_T = F(-\mathbf{I}+\alpha T)$$

The enthalpy can then be retrieved from C_p and α_p (or α) by:

$$H = H_{\mathrm{ref}} + \int_{\mathbf{r}_0}^{\mathbf{r}_1} \nabla_{\mathbf{r}} H(\mathbf{r}) \cdot d\mathbf{r} \tag{4-5}$$

where $\mathbf{r}$ is the integration vector variable, containing temperature and pressure or stress tensor components:

$$\mathbf{r} = \begin{pmatrix} p \\ T \end{pmatrix} \qquad \text{or} \qquad \mathbf{r} = \begin{pmatrix} P_{11} \\ P_{22} \\ P_{33} \\ P_{12} \\ P_{23} \\ P_{13} \\ T \end{pmatrix}$$

The starting point, $\mathbf{r}_0$, is the value of $\mathbf{r}$ at reference conditions, that is, p_{ref} (one atmosphere) and T_{ref} (298.15 K) for a fluid. The ending point, $\mathbf{r}_1$, is the solution returned after simulation. In theory any value can be assigned to the enthalpy at reference conditions, H_{ref} (Ref. 2), and COMSOL Multiphysics sets it to 0 J/kg by default. The integral in Equation 4-5 is sometimes referred to as the *sensible enthalpy* (Ref. 2) and is evaluated by numerical integration.

For the evaluation of H to work, it is important that the dependencies of C_p, ρ, and γ on the temperature are prescribed either via Model Inputs or as functions of the temperature variable. If C_p, ρ, or γ depends on the pressure, that dependency must be prescribed either via a model input or by using the variable `pA`, which is the variable for the absolute pressure in COMSOL Multiphysics.

The Physical Mechanisms under Heat Transfer

The amount of heat transferred per unit time (heat transfer rate) depends on the underlying physical mechanisms that define the mode of transfer. These are:

- *Conduction* — Heat conduction (or *diffusion*) occurs as a consequence of different mechanisms in different media. Theoretically, it takes place in a gas through collisions of molecules; in a fluid through oscillations of each molecule in a "cage" formed by its nearest neighbors; in metals mainly by electrons carrying heat and in other solids by molecular motion, which in crystals take the form of lattice vibrations known as phonons.

  In a continuous medium, Fourier's law of heat conduction states that the conductive heat flux, $\mathbf{q}$, is proportional to the temperature gradient:

  $$\mathbf{q} = -k\nabla T \qquad (4\text{-}6)$$

  The coefficient of proportionality, k, is the thermal conductivity (SI unit: W/(m·K)) and takes a positive value meaning that heat flows from regions of high temperature to low temperature. More generally, the thermal conductivity can take the form of a symmetric positive-definite second-order tensor (matrix) in anisotropic media such as composite materials:

  $$k = \begin{bmatrix} k_{xx} & k_{xy} & k_{xz} \\ k_{xy} & k_{yy} & k_{yz} \\ k_{xz} & k_{yz} & k_{zz} \end{bmatrix}$$

Thermal conductivity tensors that do not respect the symmetric positive-definite property lead to unphysical results (Ref. 3).

- *Convection* — Heat convection (sometimes called heat advection) takes place through the net displacement of a fluid that transports the heat content with its velocity. The term convection (especially convective cooling and convective heating) also refers to the heat dissipation from a solid surface to a fluid, typically described by a heat transfer coefficient.
- *Radiation* — Heat transfer by radiation takes place through the transport of photons. Participating (or semitransparent) media absorb, emit, and scatter photons. Opaque surfaces absorb or reflect them.

The Heat Balance Equation

The equations of heat transfer in continua are derived from the first law of thermodynamics, commonly referred to as the principle of conservation of energy. The present part establishes the heat balance equation in its integral and localized forms that stand as a root for deriving the different heat transfer equations solved in COMSOL Multiphysics.

INTEGRAL FORM

The first law of thermodynamics states that the variations of macroscopic kinetic energy, K_Ω, and internal energy, E_Ω, of a domain Ω are caused either by the mechanical power of forces applied to the system, P_{ext}, or by exchanged heat rate, Q_{exch} (2.3.53 in Ref. 4):

$$\frac{dE_\Omega}{dt}+\frac{dK_\Omega}{dt}=P_{\mathrm{ext}}+Q_{\mathrm{exch}} \tag{4-7}$$

Mass and momentum balance are needed to complete the description of the system. The mechanical laws, either for solids or fluids, generate the following balance equation between variation of kinetic energy, K_Ω, stress power, P_{str}, and power of applied forces, P_{ext} (2.3.64 in Ref. 4):

$$\frac{dK_\Omega}{dt}=P_{\mathrm{str}}+P_{\mathrm{ext}} \tag{4-8}$$

This equation involves quantities of the macroscopic level where the variation of the kinetic energy due to some forces applied to it reflects a sensible displacement. In COMSOL Multiphysics, the Solid Mechanics or Single-Phase Flow interfaces are examples of physics interfaces that simulate the macroscopic level described by Equation 4-8.

Combining Equation 4-7 and Equation 4-8 yields the so-called heat balance equation (2.3.65 in Ref. 4):

$$\frac{dE_\Omega}{dt} = -P_{\mathrm{str}} + Q_{\mathrm{exch}} \tag{4-9}$$

This time, the equation involves quantities of the microscopic level (exchanged heat rate, Q_{exch}, and internal energy, E_Ω) more concerned with the atomic vibrations and similar microscopic phenomena that are felt as heat. The presence of the stress power, P_{str}, in both Equation 4-8 and Equation 4-9 stands for the fact that such power is converted into heat by dissipation. The Heat Transfer interfaces, described in the next sections, simulate the heat exchanges described by Equation 4-9.

LOCALIZED FORM

In this paragraph, the different terms of Equation 4-9 are more detailed to obtain the localized form of the heat balance equation.

Variation of Internal Energy

The equations given in the previous paragraph holds for a given macroscopic continuous domain Ω where the internal energy is defined using the specific internal energy (per unit mass), E, as:

$$E_\Omega = \int_\Omega E\,dm$$

Note that by conservation of mass, the variation of internal energy in time is:

$$\frac{dE_\Omega}{dt} = \int_\Omega \frac{dE}{dt}dm = \int_\Omega \rho\frac{dE}{dt}dv$$

In these last relations, ρ is the density, and dv denotes an elementary volume of Ω. Contrary to the constant elementary mass, dm, the elementary volume changes by expansion or contraction of the domain. Recall that the derivation operator d/dt under the integrals is in the material frame (see Time Derivative in the Frames for the Heat Transfer Equations section).

Stress Power

The stress power, derived from the Continuum Mechanics theory, is defined by (2.3.59 in Ref. 4):

$$P_{\mathrm{str}} = \int_\Omega (\sigma:\mathbf{D})dv$$

where σ is the Cauchy stress tensor and $\mathbf{D}$ is the strain rate tensor. The operation ":" is a contraction and can in this case be written on the following form:

$$\mathbf{a}:\mathbf{b} = \sum_n \sum_m a_{nm} b_{nm}$$

Note that in fluid mechanics, the Cauchy stress tensor is divided into a static part for the pressure, p, and a symmetric deviatoric part, τ, as in:

$$\sigma = -p\mathbf{I} + \tau \tag{4-10}$$

so that P_{str} becomes the following sum of pressure-volume work and viscous dissipation:

$$P_{\mathrm{str}} = -\int_{\Omega} p(\nabla \cdot \mathbf{u})dv + \int_{\Omega} (\tau:\nabla\mathbf{u})dv$$

Exchanged Heat

Finally, the exchanged heat rates, Q_{exch}, account for thermal conduction (see Fourier's Law at Equation 4-6), radiation and potentially additional heat sources. Joule heating and exothermic chemical reactions are such examples of domain heat source. The different kinds of exchanged heat are summarized by the equality below:

$$Q_{\mathrm{exch}} = -\int_{\partial\Omega} (\mathbf{q} \cdot \mathbf{n})ds - \int_{\partial\Omega} (\mathbf{q}_{\mathrm{r}} \cdot \mathbf{n})ds + \int_{\Omega} Q dv$$

Recall the following notations used above: $\mathbf{q}$ for the heat flux by conduction, $\mathbf{q}_{\mathrm{r}}$ for the heat flux by radiation, Q for additional heat sources, and $\mathbf{n}$ for the external normal vector to the boundary ∂Ω.

Localized Heat Balance Equation

With all these elements, the heat balance equation (Equation 4-9) becomes:

$$\int_{\Omega} \rho \frac{dE}{dt} dv + \int_{\partial\Omega} (\mathbf{q} \cdot \mathbf{n})ds + \int_{\partial\Omega} (\mathbf{q}_{\mathrm{r}} \cdot \mathbf{n})ds = -\int_{\Omega} (\sigma:\mathbf{D})dv + \int_{\Omega} Q dv \tag{4-11}$$

which leads to the following localized form in the *material* frame:

$$\rho \frac{dE}{dt} + \nabla \cdot (\mathbf{q} + \mathbf{q}_{\mathrm{r}}) = -(\sigma:\mathbf{D}) + Q \tag{4-12}$$

or equivalently in the *spatial* frame:

$$\rho\frac{\partial E}{\partial t} + \rho\mathbf{u}\cdot\nabla E + \nabla\cdot(\mathbf{q}+\mathbf{q}_\mathrm{r}) = -(\sigma:\mathbf{D}) + Q \tag{4-13}$$

This verbally means that variations of internal energy in time are balanced by convection of internal energy, thermal conduction, radiation, dissipation of mechanical stress and additional volumetric heat sources. In the next sections, Equation 4-13 will be derived to obtain the heat transfer equations in different media.

See Frames for the Heat Transfer Equations for more details about the use of material and spatial frames in the Heat Transfer interfaces.

Consistency with Mass and Momentum Conservation Laws

Although the heat transfer interfaces only solve for the energy equation, the context leading to Equation 4-13 does account for the three additional conservation laws that complete the Continuum Mechanics theory:

- Conservation of mass
- Conservation of linear momentum
- Conservation of angular momentum

The equations corresponding to each of them are recalled below in Table 4-1. For more details about the theory of Solid and Fluid Mechanics, see the *Structural Mechanics Module User's Guide* and *CFD Module User's Guide*.

TABLE 4-1: CONSERVATION OF MASS AND MOMENTUM

CONSERVATION LAW	**MATHEMATICAL EXPRESSION IN MATERIAL FRAME**	**MATHEMATICAL EXPRESSION IN SPATIAL FRAME**
Conservation of Mass	$\rho_0 = \rho\det(\mathbf{F})$	$\frac{\partial\rho}{\partial t} + \nabla\cdot(\rho\mathbf{u}) = 0$
Conservation of Linear Momentum	$\rho\frac{d\mathbf{u}}{dt} = \nabla\cdot\sigma + \mathbf{F}_\mathrm{v}$	$\rho\frac{\partial\mathbf{u}}{\partial t} + \rho(\mathbf{u}\cdot\nabla)\mathbf{u} = \nabla\cdot\sigma + \mathbf{F}_\mathrm{v}$
Conservation of Angular Momentum	$\sigma^T = \sigma$	$\sigma^T = \sigma$

When modeling a heat transfer problem with one of the Heat Transfer interfaces, the aforementioned laws needs to be respected. For example, the velocity field, $\mathbf{u}$, provided in the energy equation and responsible for convection in a fluid, should satisfy the continuity equation here below in order to avoid unphysical results.

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

This is especially true when the velocity field is not computed from an additional physics interface — such as Single-Phase Flow that solves for the continuity equation — but instead defined by a custom expression in a Heat Transfer interface.

Theory for Heat Transfer in Solids

The Heat Transfer in Solids Interface solves for the following equation derived from Equation 4-13:

$$\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u}_{\text{trans}} \cdot \nabla T\right) + \nabla \cdot (\mathbf{q} + \mathbf{q}_\mathrm{r}) = -\alpha T{:}\frac{dS}{dt} + Q \tag{4-14}$$

The different quantities involved here are recalled below:

- ρ is the density (SI unit: $\mathrm{kg/m^3}$)
- C_p is the specific heat capacity at constant stress (SI unit: J/(kg·K))
- T is the absolute temperature (SI unit: K)
- $\mathbf{u}_{\text{trans}}$ is the velocity vector of translational motion (SI unit: m/s)
- $\mathbf{q}$ is the heat flux by conduction (SI unit: $\mathrm{W/m^2}$)
- $\mathbf{q}_\mathrm{r}$ is the heat flux by radiation (SI unit: $\mathrm{W/m^2}$)
- α is the coefficient of thermal expansion (SI unit: 1/K)
- S is the second Piola-Kirchhoff stress tensor (SI unit: Pa)
- Q contains additional heat sources (SI unit: $\mathrm{W/m^3}$)

For a steady-state problem the temperature does not change with time and the terms with time derivatives disappear.

The first term on the right-hand side of Equation 4-14 is the *thermoelastic damping* and accounts for thermoelastic effects in solids:

$$Q_{\text{ted}} = -\alpha T{:}\frac{dS}{dt} \tag{4-15}$$

It should be noted that the d/dt operator is the material derivative, as described in the Time Derivative subsection of Material and Spatial Frames.

Theory for Heat Transfer in Fluids

The Heat Transfer in Fluids Interface solves for the following equation (11.2-5 in Ref. 5):

$$\rho C_p\left(\frac{\partial T}{\partial t}+\mathbf{u}\cdot\nabla T\right)+\nabla\cdot(\mathbf{q}+\mathbf{q}_\mathrm{r}) = \alpha_p T\left(\frac{\partial p}{\partial t}+\mathbf{u}\cdot\nabla p\right)+\tau:\nabla\mathbf{u}+Q \qquad (4\text{-}16)$$

which is derived from Equation 4-13, considering that:

- the Cauchy stress tensor, σ, is split into static and deviatoric parts as in:

$$\sigma = -p\mathbf{I}+\tau$$

- the dependent variables are the temperature, T, and pressure, p.

The different quantities involved here are recalled below:

- ρ is the density (SI unit: $\mathrm{kg/m^3}$)
- C_p is the specific heat capacity at constant pressure (SI unit: J/(kg·K))
- T is the absolute temperature (SI unit: K)
- $\mathbf{u}$ is the velocity vector (SI unit: m/s)
- $\mathbf{q}$ is the heat flux by conduction (SI unit: $\mathrm{W/m^2}$)
- $\mathbf{q}_\mathrm{r}$ is the heat flux by radiation (SI unit: $\mathrm{W/m^2}$)
- α_p is the coefficient of thermal expansion (SI unit: 1/K):

$$\alpha_p = -\frac{1}{\rho}\frac{\partial\rho}{\partial T}$$

  for ideal gases, the thermal expansion coefficient takes the simpler form $\alpha_p = 1/T$

- p is the pressure (SI unit: Pa)
- τ is the viscous stress tensor (SI unit: Pa)
- Q contains heat sources other than viscous dissipation (SI unit: $\mathrm{W/m^3}$)

For a steady-state problem the temperature does not change with time and the terms with time derivatives disappear.

The first term of the right-hand side of Equation 4-16 is the *work done by pressure changes* and is the result of heating under adiabatic compression as well as some thermoacoustic effects. It is generally small for low Mach number flows.

$$Q_p = \alpha_p T\left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p\right) \tag{4-17}$$

The second term represents viscous dissipation in the fluid:

$$Q_{\mathrm{vd}} = \tau{:}\nabla\mathbf{u} \tag{4-18}$$

Theory for Bioheat Transfer

The Bioheat Equation

The Bioheat Transfer Interface solves for the bioheat equation using Pennes' approximation. This is used to model heat transfer within biological tissue. It accounts for heat sources from blood perfusion and metabolism in the classical heat transfer equation:

$$\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = \rho_\mathrm{b} C_{p,\,\mathrm{b}} \omega_\mathrm{b} (T_\mathrm{b} - T) + Q_\mathrm{met} \tag{4-19}$$

The different quantities involved here are recalled below:

- ρ is the density of the tissue (SI unit: $\mathrm{kg/m^3}$)
- C_p is the specific heat capacity at constant pressure of the tissue (SI unit: $\mathrm{J/(kg{\cdot}K)}$)
- T is the absolute temperature of the tissue (SI unit: K)
- $\mathbf{q}$ is the heat flux by conduction in the tissue (SI unit: $\mathrm{W/m^2}$)
- ρ_b is the blood density (SI unit: $\mathrm{kg/m^3}$)
- $C_{p,\,\mathrm{b}}$ is the blood specific heat capacity at constant pressure (SI unit: $\mathrm{J/(kg{\cdot}K)}$)
- ω_b is the blood perfusion rate (SI unit: $\mathrm{1/s}$)
- T_b is the arterial blood temperature (SI unit: K)
- Q_met is the metabolic heat source (SI unit: $\mathrm{W/m^3}$)

For a steady-state problem the temperature does not change with time and the terms with time derivatives disappear.

There are specific predefined materials available in the Bioheat material database. See Materials Overview and Bioheat Material Database in the *COMSOL Multiphysics Reference Manual*.

Damaged Tissue

On the Biological Tissue node, select the **Include damage integral analysis** check box to calculate tissue damage.

In hyperthermia and cryogenic processes, tissue necrosis (permanent damage or death of living tissue) occurs when one of the two phenomenas happens:

- an excessive thermal energy is absorbed (hyperthermia process) or released (cryogenic process),
- a critical high (hyperthermia process) or low (cryogenic process) temperature is exceeded.

Correspondingly, COMSOL Multiphysics has two ways to model energy absorption — computing the period of time the tissue remained in the necrotic temperature interval and direct time integration of the energy.

First Form Integral

In the first form of damage integral, tissue necrosis occurs in four cases:

- When the temperature exceeds the hyperthermia damage temperature $T_{\mathrm{d,\,h}}$ for more than a certain time period $t_{\mathrm{d,\,h}}$,
- When the temperature falls below the cryogenic damage temperature $T_{\mathrm{d,\,c}}$ for more than a certain time period $t_{\mathrm{d,\,c}}$,
- Instantly after the temperature exceeds the hyperthermia necrosis temperature $T_{\mathrm{n,\,h}}$,
- Instantly after the temperature falls below the cryogenic necrosis temperature $T_{\mathrm{n,\,c}}$.

For the first two cases, the damaged tissue indicator, α_1, defined by

$$\alpha_1 = \frac{1}{t_{\mathrm{d,\,h}}}\int_0^t \varphi_{\mathrm{d,\,h}}\,dt + \frac{1}{t_{\mathrm{d,\,c}}}\int_0^t \varphi_{\mathrm{d,\,c}}\,dt$$

$$\varphi_{\mathrm{d,\,h}}(t) = \begin{cases} 1 & \text{if } T > T_{\mathrm{d,\,h}} \\ 0 & \text{otherwise} \end{cases} \qquad \varphi_{\mathrm{d,\,c}}(t) = \begin{cases} 1 & \text{if } T < T_{\mathrm{d,\,c}} \\ 0 & \text{otherwise} \end{cases}$$

is the ratio of the period of time when $T > T_{\mathrm{d,\,h}}$ to the time limit $t_{\mathrm{d,\,h}}$, plus the ratio of the period of time when $T < T_{\mathrm{d,\,c}}$ to the time limit $t_{\mathrm{d,\,c}}$. It gives an indication of damage state of the tissue. When it reaches 1, the tissue is necrotic. The fraction of necrotic tissue corresponds to the quantity $\min(\alpha_1, 1)$.

For the last two cases, the necrosis time indicator, α_2, defined by

$$\alpha_2 = \int_0^t \varphi_{\mathrm{n,\,h}}\,dt + \int_0^t \varphi_{\mathrm{n,\,c}}\,dt$$

$$\varphi_{\mathrm{n,\,h}}(t) = \begin{cases} 1 & \text{if } T > T_{\mathrm{n,\,h}} \\ 0 & \text{otherwise} \end{cases} \qquad \varphi_{\mathrm{n,\,c}}(t) = \begin{cases} 1 & \text{if } T < T_{\mathrm{n,\,c}} \\ 0 & \text{otherwise} \end{cases}$$

evaluates the period of time when $T > T_{\mathrm{n,\,h}}$ plus the period of time when $T < T_{\mathrm{n,\,c}}$. If $\alpha_2 > 0$, the tissue is necrotic because it already reached the necrosis temperatures $T_{\mathrm{n,\,h}}$ or $T_{\mathrm{n,\,c}}$ at some time step of the simulation. Hence, the fraction of necrotic tissue due to immediate necrosis is equal to 1 if $\alpha_2 > 0$ and 0 otherwise.

Combining all cases, the overall fraction of necrotic tissue, θ_{d}, is equal to:

$$\theta_{\mathrm{d}} = \begin{cases} 1 & \text{if } \alpha_2 > 0 \\ \min(\alpha_1, 1) & \text{otherwise} \end{cases} \tag{4-20}$$

Second Form Integral

The second form of damage integral is applicable only for hyperthermia processes and provides the degree of tissue injury, α, based on the Arrhenius equation:

$$\alpha = \int_0^t A e^{\frac{-\Delta E}{RT}} dt$$

Here, A is the frequency factor (SI unit: 1/s), and ΔE is the activation energy for the irreversible damage reaction (SI unit: J/mol). The parameters A and ΔE are dependent on the type of tissue and have been characterized for liver tissues by Jacques et others (Ref. 6) to be $A = 7.39 \cdot 10^{39}\ \mathrm{s}^{-1}$ and $\Delta E = 2.577 \cdot 10^5$ J/mol. See Ref. 7, Ref. 8, and Ref. 9 for the characterization of these parameters for prostate, skin, and fat. See also Ref. 10 and Ref. 11 for more references on biological tissues material properties.

The fraction of necrotic tissue is then expressed by:

$$\theta_{\mathrm{d}} = 1 - e^{-\alpha} \tag{4-21}$$

Thermal Properties

The material properties of the damaged tissue is redefined to take into account the influence of tissue injury. If ρ_{d}, $C_{p,\,\mathrm{d}}$, and k_{d} denote the density, heat capacity at constant pressure, and thermal conductivity of the necrotic tissue, respectively, then two effective quantities are defined:

- The effective thermal conductivity, $k_{\mathrm{eff}} = \theta_{\mathrm{d}} k_{\mathrm{d}} + (1 - \theta_{\mathrm{d}})k$
- The effective heat capacity at constant pressure, $(\rho C_p)_{\mathrm{eff}} = \theta_{\mathrm{d}} \rho_{\mathrm{d}} C_{p,\,\mathrm{d}} + (1 - \theta_{\mathrm{d}})\rho C_p$

In these equalities, θ_d takes one of the two definitions given above in Equation 4-20 or Equation 4-21 according to the integral form chosen.

Theory for Heat Transfer in Porous Media

The heat transfer equation for porous media is derived from the mixture rule on energies appearing in solid and fluid heat transfer equations (see Ref. 12). For undeformed immobile solids, Equation 4-14 simplifies into:

$$\rho_s C_{p,s} \frac{\partial T_s}{\partial t} + \nabla \cdot \mathbf{q}_s = Q_s$$

and for a fluid domain where pressure work and viscous dissipation are neglected, Equation 4-16 becomes:

$$\rho_f C_{p,f} \frac{\partial T_f}{\partial t} + \rho_f C_{p,f} \mathbf{u}_f \cdot \nabla T_f + \nabla \cdot \mathbf{q}_f = Q_f$$

The mixture rule applies by multiplying the first equation by the solid volume fraction, θ_p, multiplying the second one by the porosity, $1-\theta_p$, and summing resulting equations.

The local thermal equilibrium hypothesis assumes equality of temperature in both fluid and solid phases:

$$T_f = T_s = T \tag{4-22}$$

The theory for this hypothesis is detailed in the section Local Thermal Equilibrium below. Otherwise, the Local Thermal Non-Equilibrium section describes the theory for modeling heat transfer in porous media using two temperatures.

When Should Local Thermal Equilibrium and Non-Equilibrium be Considered?

The classical local equilibrium hypothesis in modeling heat transfer in porous media considers pointwise equality of solid and fluid temperatures as said in Equation 4-22. The Local Thermal Equilibrium section below details the derivation of the energy equation considering such assumption that remains accurately sufficient for several applications. Ref. 27 shows for instance that solid and fluid temperatures are equal in steady conduction problems where only prescribed temperature conditions are applied. Most slow motion problems can also assume equality of phase temperatures if volumetric internal heating do not differ in both materials.

In the case of conduction in porous plates, Ref. 28 provides criteria based on the dimensionless Sparrow number, Sp, to indicate if temperature equilibrium is still valid or if a non-equilibrium point of view should be preferred. In Ref. 29, the influence of the Darcy number, Da, and the ratio of phase conductivities is examined for transient heat transfer in packed beds. The Sparrow and Darcy numbers are defined by:

$$\mathrm{Sp} = \frac{h_{\mathrm{sf}}L^2}{k_{\mathrm{eff}}r_{\mathrm{h}}} \qquad \mathrm{Da} = \frac{\kappa}{d^2}$$

where:

- h_{sf} is the interstitial heat transfer coefficient between solid and fluid phases (SI unit: $\mathrm{W/(m^2{\cdot}K)}$)
- L is the plate layer thickness (SI unit: m)
- k_{eff} is the equivalent thermal conductivity of the porous medium (SI unit: $\mathrm{W/(m{\cdot}K)}$)
- r_{h} is the hydraulic radius (SI unit: m)
- κ is the permeability (SI unit: $\mathrm{m^2}$)
- d is the average particle diameter (SI unit: m)

In the situations described in Ref. 28 and Ref. 29, small values of Sp (less than 100 or 500) and large values of Da (from order of magnitude 10^{-7}) indicate discrepancies of temperature in each phase. However, in general, assessing the validity of local thermal equilibrium assumption remains not straightforward in specific situations. The Local Thermal Non-Equilibrium approach, described below, makes use of two energy equations, one for each phase of the porous medium, that solve for two temperature fields. It numerically doubles the number of freedom to solve but provides a general frame for heat transfer in porous media where evaluating the validity of the equilibrium hypothesis is not required anymore.

Local Thermal Equilibrium

The local thermal equilibrium hypothesis of Equation 4-22 implies a common temperature, T, for both solid and fluid phase. The Heat Transfer in Porous Media Interface solves for the following version of the heat equation (Ref. 14), reformulated using T:

$$(\rho C_p)_{\text{eff}}\frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \tag{4-23}$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \tag{4-24}$$

The different quantities appearing here are:

- ρ is the fluid density.
- C_p is the fluid heat capacity at constant pressure.
- $(\rho C_p)_{\text{eff}}$ is the effective volumetric heat capacity at constant pressure, defined by

$$(\rho C_p)_{\text{eff}} = \theta_{\text{p}}\rho_{\text{p}}C_{p,\,\text{p}} + (1-\theta_{\text{p}})\rho C_p$$

- k_{eff} is the effective thermal conductivity (a scalar or a tensor if the thermal conductivity is anisotropic).
- $\mathbf{q}$ is the conductive heat flux.
- $\mathbf{u}$ is the velocity field, either an analytic expression or computed from a Fluid Flow interface. It should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area. The average linear velocity (the velocity within the pores) can be calculated as $\mathbf{u}_{\text{f}} = \mathbf{u}/(1-\theta_{\text{p}})$, where $(1-\theta_{\text{p}})$ is the fluid's volume fraction, or equivalently the porosity.
- Q is the heat source (or sink). Add one or several heat sources as separate physics features.

For a steady-state problem the temperature does not change with time, and the terms with time derivatives of Equation 4-23 disappear.

The effective thermal conductivity of the solid-fluid system, k_{eff}, is related to the conductivity of the solid, k_{p}, and to the conductivity of the fluid, k, and depends in a complex way on the geometry of the medium. In Ref. 12, three models are proposed for an isotropic medium:

- If the heat conduction occurs in parallel in the solid and the fluid, then the effective thermal conductivity is the weighted arithmetic mean of the conductivities k and k_{p}:

$$k_{\text{eff}} = \theta_{\text{p}}k_{\text{p}} + (1-\theta_{\text{p}})k$$

  This volume average model provides an upper bound for the effective thermal conductivity.

- If the heat conduction takes place in series, with all of the heat flux passing through both solid and fluid, then the effective thermal conductivity is the weighted harmonic mean of the conductivities k and k_p:

$$\frac{1}{k_\mathrm{eff}} = \frac{\theta_\mathrm{p}}{k_\mathrm{p}} + \frac{(1-\theta_\mathrm{p})}{k}$$

  This reciprocal average model provides a lower bound for the effective thermal conductivity.

- A last estimate is given by the weighted geometric mean of k and k_p:

$$k_\mathrm{eff} = k_\mathrm{p}^{\theta_\mathrm{p}} \cdot k^{1-\theta_\mathrm{p}}$$

  This model provides a good estimate as long as k and k_p are not too different from each other.

When k and k_p are equal the three models give the same effective thermal conductivity.

Local Thermal Non-Equilibrium

The Local Thermal Non-Equilibrium Interface implements heat transfer in porous media for which the temperatures into the porous matrix and the fluid are not in equilibrium.

EQUATIONS FOR LOCAL NON-EQUILIBRIUM HEAT TRANSFER

Non-equilibrium heat transfer in porous media for binary systems of rigid porous matrix and fluid phase are governed by a set of two equations. These are the usual heat equations for solids and fluids, multiplied by the volume fractions θ_p and $(1-\theta_\mathrm{p})$ respectively, and with an additional source term quantifying exchanged heat between both phases (2.12 and 2.13 in Ref. 12):

$$\theta_\mathrm{p}\rho_\mathrm{s}C_{p,\,\mathrm{s}}\frac{\partial T_\mathrm{s}}{\partial t} + \nabla\cdot\mathbf{q}_\mathrm{s} = q_\mathrm{sf}(T_\mathrm{f}-T_\mathrm{s}) + \theta_\mathrm{p}Q_\mathrm{s} \tag{4-25}$$

$$\mathbf{q}_\mathrm{s} = -\theta_\mathrm{p}k_\mathrm{s}\nabla T_\mathrm{s}$$

$$(1-\theta_\mathrm{p})\rho_\mathrm{f}C_{p,\,\mathrm{f}}\frac{\partial T_\mathrm{f}}{\partial t} + (1-\theta_\mathrm{p})\rho_\mathrm{f}C_{p,\,\mathrm{f}}\mathbf{u}_\mathrm{f}\cdot\nabla T_\mathrm{f} + \nabla\cdot\mathbf{q}_\mathrm{f} = q_\mathrm{sf}(T_\mathrm{s}-T_\mathrm{f}) + (1-\theta_\mathrm{p})Q_\mathrm{f}$$

$$\mathbf{q}_\mathrm{f} = -(1-\theta_\mathrm{p})k_\mathrm{f}\nabla T_\mathrm{f}$$

In these expressions:

- θ_p is the solid volume fraction (SI unit: 1)
- ρ_s and ρ_f are the solid and fluid densities (SI unit: kg/m^3)
- $C_{p,s}$ and $C_{p,f}$ are the solid and fluid heat capacities at constant pressure (SI unit: J/(kg·K))
- $\mathbf{q}_s$ and $\mathbf{q}_f$ are the solid and fluid conductive heat fluxes (SI unit: W/m^2)
- k_s and k_f are the solid and fluid thermal conductivities (SI unit: W/(m·K))
- q_{sf} is the interstitial convective heat transfer coefficient (SI unit: W/(m^3·K))
- Q_s and Q_f are the solid and fluid heat sources (SI unit: W/m^3)
- $\mathbf{u}_f$ is the fluid velocity vector (SI unit: m/s)

The fluid velocity is often deduced from a porous velocity $\mathbf{u}_p$, coming for example from Darcy's law or Brinkman equations, according to:

$$\mathbf{u}_f = \frac{\mathbf{u}_p}{1-\theta_p}$$

so that the heat equations in the fluid domain reduces to:

$$(1-\theta_p)\rho_f C_{p,f}\frac{\partial T_f}{\partial t} + \rho_f C_{p,f}\mathbf{u}_p \cdot \nabla T_f + \nabla \cdot \mathbf{q}_f = q_{sf}(T_s - T_f) + (1-\theta_p)Q_f \qquad (4\text{-}26)$$

$$\mathbf{q}_f = -(1-\theta_p)k_f \nabla T_f$$

The Local Thermal Non-Equilibrium multiphysics coupling adds the exchanged opposite heat sources $q_{sf}(T_f - T_s)$ and $q_{sf}(T_s - T_f)$ that one phase receives from or releases to the other when respective temperatures differ. The porous temperature, T, has the following definition (Ref. 30):

$$T = \frac{\theta_p\rho_s C_{p,s}T_s + (1-\theta_p)\rho_f C_{p,f}T_f}{\theta_p\rho_s C_{p,s} + (1-\theta_p)\rho_f C_{p,f}}$$

CORRELATION FOR THE INTERSTITIAL CONVECTIVE HEAT TRANSFER COEFFICIENT

The Local Thermal Non-Equilibrium multiphysics feature provides a built-in correlation for q_{sf} in the spherical pellet bed configuration (2.14, 2.15, and 2.16 in Ref. 12):

$$q_{sf} = a_{sf}h_{sf}$$

The specific surface area, a_{sf} (SI unit: 1/m), for a bed packed with spherical particles of radius r_{p} is:

$$a_{\mathrm{sf}} = \frac{6\theta_{\mathrm{p}}}{2r_{\mathrm{p}}}$$

The interstitial heat transfer coefficient, h_{sf} (SI unit: W/(m^2·K)), satisfies the relation:

$$\frac{1}{h_{\mathrm{sf}}} = \frac{2r_{\mathrm{p}}}{k_{\mathrm{f}}\mathrm{Nu}} + \frac{2r_{\mathrm{p}}}{\beta k_{\mathrm{s}}}$$

where $\beta = 10$ for spherical particles, and Nu is the fluid-to-solid Nusselt number derived from following correlation (2.17 in Ref. 12):

$$\mathrm{Nu} = 2.0 + 1.1\mathrm{Pr}^{1/3}\mathrm{Re}_{\mathrm{p}}^{0.6}$$

The Prandtl number, Pr, and particle Reynolds number, Re_{p}, are defined by:

$$\mathrm{Pr} = \frac{\mu C_{p,\mathrm{f}}}{k_{\mathrm{f}}} \qquad \mathrm{Re}_{\mathrm{p}} = \frac{2r_{\mathrm{p}}\rho_{\mathrm{p}}\|\mathbf{u}_{\mathrm{f}}\|}{\mu}$$

VOLUMETRIC AND SURFACE THERMAL CONDITIONS

Because the Local Thermal Non-Equilibrium multiphysics coupling multiplies each energy equation by its volume fraction, θ_{p} and $(1 - \theta_{\mathrm{p}})$ for solid and fluid phases respectively, a heat source or heat flux defined in a couple heat transfer interface is also accounted with that ratio. As shown in Equation 4-25 and Equation 4-26, the volumetric heat sources $\theta_{\mathrm{p}}Q_{\mathrm{s}}$ and $(1 - \theta_{\mathrm{p}})Q_{\mathrm{f}}$ are applied to the energy equations while the Heat Source features of each physics interface specify Q_{s} and Q_{f}.

Theory for Heat Transfer with Phase Change

The Phase Change Material node is used to solve the heat equation after specifying the properties of a phase change material according to the *apparent heat capacity* formulation.

Instead of adding a latent heat L in the energy balance equation exactly when the material reaches its phase change temperature T_{pc}, it is assumed that the transformation occurs in a temperature interval between $T_{\mathrm{pc}} - \Delta T/2$ and $T_{\mathrm{pc}} + \Delta T/2$. In this interval, the material phase is modeled by a smoothed function, θ, representing the fraction of phase before transition, which is equal to 1 before $T_{\mathrm{pc}} - \Delta T/2$ and to 0 after $T_{\mathrm{pc}} + \Delta T/2$. The density, ρ, and the specific enthalpy, H, are expressed by:

$$\rho = \theta\rho_{\mathrm{ph1}} + (1-\theta)\rho_{\mathrm{ph2}}$$

$$H = \frac{1}{\rho}(\theta\rho_{\mathrm{ph1}}H_{\mathrm{ph1}} + (1-\theta)\rho_{\mathrm{ph2}}H_{\mathrm{ph2}})$$

where the indices ph1 and ph2 indicate a material in phase 1 or in phase 2, respectively. Differentiating with respect to temperature, this equality provides the following formula for the specific heat capacity:

$$C_p = \frac{\partial H}{\partial T}$$

which becomes, after some formal transformations:

$$C_p = \frac{1}{\rho}(\theta_1\rho_{\mathrm{ph1}}C_{p,\,\mathrm{ph1}} + \theta_2\rho_{\mathrm{ph2}}C_{p,\,\mathrm{ph2}}) + (H_{\mathrm{ph2}} - H_{\mathrm{ph1}})\frac{d\alpha_{\mathrm{m}}}{dT}$$

Here, θ_1 and θ_2 are equal to θ and $1-\theta$, respectively. The mass fraction, α_{m}, is defined from ρ_{ph1}, ρ_{ph2} and θ according to:

$$\alpha_{\mathrm{m}} = \frac{1}{2}\frac{\theta_2\rho_{\mathrm{ph2}} - \theta_1\rho_{\mathrm{ph1}}}{\rho}$$

It is equal to $-1/2$ before transformation and $1/2$ after transformation. The specific heat capacity is the sum of an equivalent heat capacity C_{eq}:

$$C_{\mathrm{eq}} = \frac{1}{\rho}(\theta_1\rho_{\mathrm{ph1}}C_{p,\,\mathrm{ph1}} + \theta_2\rho_{\mathrm{ph2}}C_{p,\,\mathrm{ph2}})$$

and the distribution of latent heat C_L:

$$C_L(T) = (H_{\mathrm{ph2}} - H_{\mathrm{ph1}})\frac{d\alpha_{\mathrm{m}}}{dT}$$

In the ideal case, when $1 - \theta$ is the Heaviside function (equal to 0 before T_{pc} and to 1 after T_{pc}), $d\alpha_{\mathrm{m}}/dT$ is the Dirac pulse.

Therefore, C_L is the enthalpy jump, L, at temperature T_{pc} that is added when you have a pure substance.

The latent heat distribution C_L is approximated by

$$C_L(T) = L\frac{d\alpha_{\mathrm{m}}}{dT}$$

so that the total heat per unit volume released during the phase transformation coincides with the latent heat:

$$\int_{T_{\mathrm{pc}} - \frac{\Delta T}{2}}^{T_{\mathrm{pc}} + \frac{\Delta T}{2}} C_L(T)dT = L\int_{T_{\mathrm{pc}} - \frac{\Delta T}{2}}^{T_{\mathrm{pc}} + \frac{\Delta T}{2}} \frac{d\alpha_{\mathrm{m}}}{dT}dT = L$$

The latent heat, L, can depend on the absolute pressure but should not depend on the temperature.

Finally, the apparent heat capacity, C_p, used in the heat equation, is given by:

$$C_p = \frac{1}{\rho}(\theta_1\rho_{\mathrm{ph1}}C_{p,\,\mathrm{ph1}} + \theta_2\rho_{\mathrm{ph2}}C_{p,\,\mathrm{ph2}}) + C_L$$

The effective thermal conductivity reduces to:

$$k = \theta_1 k_{\mathrm{ph1}} + \theta_2 k_{\mathrm{ph2}}$$

and the effective density is:

$$\rho = \theta_1\rho_{\mathrm{ph1}} + \theta_2\rho_{\mathrm{ph2}}$$

To satisfy energy and mass conservation in phase change models, particular attention should be paid to the density in time simulations. When the material density is not constant over time, for example, dependent on the temperature, volume change is expected. The transport velocity field and the density must be defined so that mass is conserved locally.

Moving Mesh Interface (described in the *COMSOL Multiphysics Reference Manual*) can be used to account for model deformation.

Theory for Heat Transfer in Building Materials

The Heat Transfer in Building Materials Interface solves for the following equations derived from Ref. 13:

$$(\rho C_p)_{\mathrm{eff}}\frac{\partial T}{\partial t}+\nabla\cdot\mathbf{q} = Q \tag{4-27}$$

$$\mathbf{q} = -(k_{\mathrm{eff}}\nabla T+L_{\mathrm{v}}\delta_{\mathrm{p}}\nabla(\phi p_{\mathrm{sat}})) \tag{4-28}$$

which is derived from Equation 4-13, considering the building material as a porous medium in local thermal equilibrium in which the following mixing rules apply:

- $(\rho C_p)_{\mathrm{eff}}$ (SI unit: $\mathrm{J/(m^3\cdot K)}$) is the effective volumetric heat capacity at constant pressure, defined to account for both solid matrix and moisture properties:

  $$(\rho C_p)_{\mathrm{eff}} = \rho_{\mathrm{s}}C_{p,\,\mathrm{s}}+wC_{p,\,\mathrm{w}}$$

  where ρ_{s} (SI unit: $\mathrm{kg/m^3}$) is the dry solid density, $C_{p,\mathrm{s}}$ (SI unit: $\mathrm{J/(kg\cdot K)}$) is the dry solid specific heat capacity, w (SI unit: $\mathrm{kg/m^3}$) is the water content given by a moisture storage function, and $C_{p,\mathrm{w}}$ (SI unit: $\mathrm{J/(kg\cdot K)}$) is the water heat capacity at constant pressure.
- k_{eff} (SI unit: $\mathrm{W/(m\cdot K)}$) is the effective thermal conductivity, defined in function of the solid matrix and moisture properties:

  $$k_{\mathrm{eff}} = k_{\mathrm{s}}\left(1+\frac{bw}{\rho_{\mathrm{s}}}\right)$$

  where k_{s} (SI unit: $\mathrm{W/(m\cdot K)}$) is the dry solid thermal conductivity and b (dimensionless) is the thermal conductivity supplement.

  This definition neglects the contribution due to the volume fraction change of the moist air.

The heat source due to moisture content variation is expressed as the vapor diffusion flow multiplied by latent heat of evaporation:

$$L_{\mathrm{v}}\delta_{\mathrm{p}}\nabla(\phi p_{\mathrm{sat}})$$

where L_v (SI unit: J/kg) is the latent heat of evaporation, δ_p (SI unit: s) is the vapor permeability, ϕ (dimensionless) is the relative humidity, and p_{sat} (SI unit: Pa) is the vapor saturation pressure.

Theory for Lumped Isothermal Domain

The Isothermal Domain feature considers the temperature to be homogeneous in space but not necessarily in time. This is an approximation adapted to situations where a domain is nearly at the average temperature and with small fluctuations, for instance, solid objects made of conductive material immersed in water, or global temperature of a heated and well insulated room adjacent to a cold environment.

Recalling Equation 4-16 given previously in the Theory for Heat Transfer in Fluids section, without pressure-volume work and viscous dissipation, the equation to be solved reduces to:

$$\rho C_p \frac{dT}{dt} + \nabla \cdot \mathbf{q} = Q$$

Integrating this equation over the domain leads to:

$$m\overline{C_p}\frac{dT}{dt} + \int_S (\mathbf{n} \cdot \mathbf{q})ds = \int_V Q dv \tag{4-29}$$

where the domain mass and the heat capacity at constant pressure are

$$m = \int_V \rho dv \qquad \overline{C_p} = \frac{1}{m}\int_V \rho C_p dv$$

The exterior boundaries of each Isothermal Domain need the heat exchange to be specified.

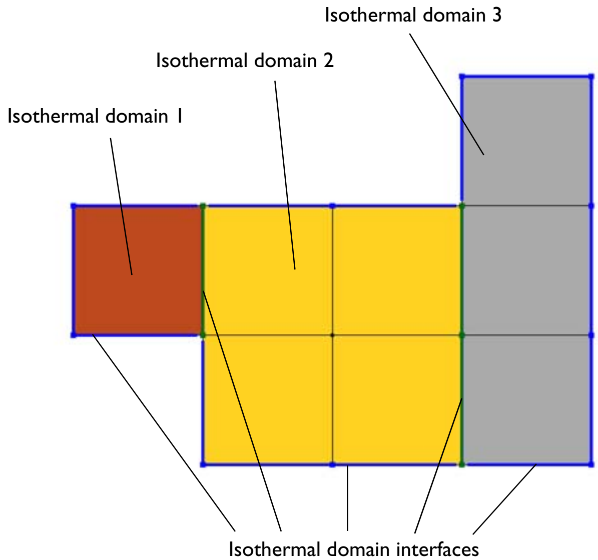


Figure 4-1: Interfaces at exterior boundaries of each Isothermal Domain.

COMSOL Multiphysics provides several types of interfaces: thermal insulation, continuity, ventilation, convective heat flux, and thermal contact.

THERMAL INSULATION

The Thermal insulation condition prevents any heat transfer between both adjacent domains.

CONTINUITY

The Continuity condition ensures equal temperature at both sides of the interface.

VENTILATION

The Ventilation condition is used for cases when an isothermal domain is considered fluid and has an adjacent domain containing the same fluid. An opening lets the fluid going from one domain to another with a determined mass flux, denoted by $\varphi_{\mathrm{d} \to \mathrm{u}}$ or $\varphi_{\mathrm{u} \to \mathrm{d}}$, respectively, along or opposite to the geometrical normal vector. The Ventilation condition is written

$$-\mathbf{n}_{\mathrm{d}} \cdot \mathbf{q}_{\mathrm{d}} = \varphi_{\mathrm{d} \to \mathrm{u}} H_{\mathrm{d}} - \varphi_{\mathrm{u} \to \mathrm{d}} H_{\mathrm{u}} \tag{4-30}$$

CONVECTIVE HEAT FLUX

The Convective heat flux condition is adapted to cases when an isothermal domain is considered solid and is adjacent to a fluid. Convection occurs at the interface with a specified heat transfer coefficient, h. The interface condition reads

$$-\mathbf{n}_{\mathrm{d}} \cdot \mathbf{q}_{\mathrm{d}} = -h(T_{\mathrm{u}} - T_{\mathrm{d}}) \tag{4-31}$$

THERMAL CONTACT

When an isothermal domain is considered solid and is adjacent to another solid, thermal contact occurs and is characterized by a given thermal resistance, R_{t}. At the interface, the condition Thermal contact reads

$$-\mathbf{n}_{\mathrm{d}} \cdot \mathbf{q}_{\mathrm{d}} = -\frac{T_{\mathrm{u}} - T_{\mathrm{d}}}{R_{\mathrm{t}}} \tag{4-32}$$

Theory for Heat Transfer in Thin Structures

In COMSOL Multiphysics, thin domains of solid, fluid or porous materials have dedicated tools to model them with boundaries instead of full domains. This way, simulations directly benefit from a reduced number of mesh elements. This section presents the hypotheses needed for such approximations and the resulting heat transfer equations that hold.

In this section:

Modeling Thin Structures

The Heat Transfer interfaces contain several lumped conditions for modeling heat transfer in thin domains: Thin Layer, Thin Film, Fracture, and Thin Rod.

In addition, standalone physics interface are available for the modeling of heat transfer by conduction, convection and radiation in thin structures:

- The Heat Transfer in Thin Shells Interface
- The Heat Transfer in Thin Films Interface
- The Heat Transfer in Fractures Interface

Either the Thin Conductive Layer (Heat Transfer in Thin Shells Interface), Thin Film, or Fracture feature is available by default in each of these interfaces. The Thin Layered Shell (Heat Transfer in Thin Shells Interface) feature is also available to model multi-layered structures.

The features mentioned above are the counterparts of domain features for the modeling of heat transfer in solid, fluid, and porous thin structures that can be represented as boundaries or edges, as described in Table 4-2.

TABLE 4-2: EQUIVALENT DOMAIN AND THIN STRUCTURES FEATURES

TYPE OF MEDIUM	DOMAIN FEATURE	BOUNDARY FEATURE	EDGE FEATURE
Solid	Solid	Thin Layer (ht) Thin Conductive Layer (ht, htsh)	Thin Rod (ht)
Fluid	Fluid	Thin Film (ht, htsh)	—
Porous	Porous Medium	Fracture (ht, htsh)	—

All these functionalities have in common the fact that the thin domains they model are lumped into boundaries (for The Heat Transfer in Thin Shells Interface, Thin Layer, Thin Film and Fracture) or 3D edges (for Thin Rod).

REDUCED MESH ELEMENT NUMBER

A significant benefit is that a thin structure can be represented as a boundary instead of a domain and a rod can be represented as a 3D edge. This simplifies the geometry and reduces the required number of mesh elements. Figure 4-2 shows an example where a thin structure significantly reduces the mesh density.

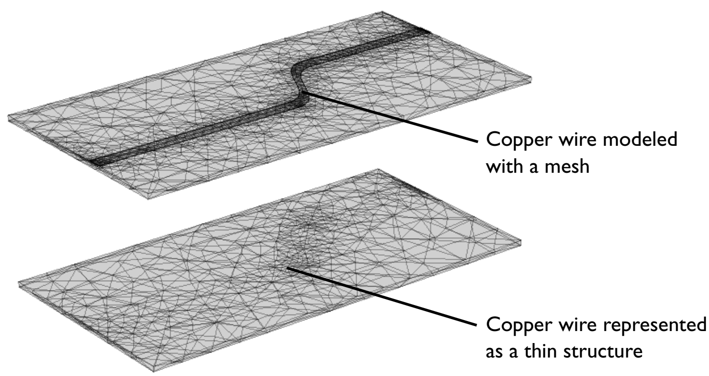


Figure 4-2: Modeling a copper wire as a domain (top) requires a denser mesh compared to modeling it as a boundary with a conductive layer (bottom).

THIN STRUCTURE AS AN EXTRA DIMENSION

To model heat transfer through the thickness of a thin structure, or multiple sandwiched layers with different material properties and thicknesses, COMSOL Multiphysics gives the possibility to create a product space between the dimensions of the boundary and an additional dimension. This is realized by the Extra Dimension tool through either the Thin Layered Shell (Heat Transfer in Thin Shells Interface) feature or the **General** option of the Thin Layer and Thin Film features.

Adding Extra Dimensions to a Model and Using Extra Dimensions in the *COMSOL Multiphysics Reference Manual.*

Plotting Results in Thin Layers Extra Dimensions

An additional 1D segmented line represents the thickness of the thin structure. The number of mesh points for each interval of the extra dimension is set to 2 by default.

TANGENTIAL AND NORMAL GRADIENTS

In thin structures, the tangential gradient and the normal gradient can be more appropriate to express the governing equations.

The *normal gradient* is the projection of the gradient operator onto the normal vector, n, of the boundary representing the thin structure. This is mathematically expressed for any scalar field T as:

$$\nabla_{\mathbf{n}} T = (\nabla T \cdot \mathbf{n})\mathbf{n}$$

The *tangential gradient* removes the normal component from the gradient operation, so that only tangential components remain. This is mathematically expressed for any scalar field T as:

$$\nabla_{\mathbf{t}} T = \nabla T - (\nabla T \cdot \mathbf{n})\mathbf{n}$$

The gradient operator is then split into a tangential part and a normal part:

$$\nabla T = \nabla_{\mathbf{t}} T + \nabla_{\mathbf{n}} T \tag{4-33}$$

This relation simplifies to:

$$\nabla T = \nabla_{\mathbf{t}} T$$

or

$$\nabla T = \nabla_{\mathbf{n}} T$$

when tangential heat transfer is dominant or negligible. These results will be useful in the next sections describing heat transfer in the different thin structures.

It should be noted that when an extra dimension is used, the equations are written from the point of view of the extra dimension. In particular, the `dtang()` operator would correspond to $\nabla_{\mathbf{n}}$ since it performs the derivation along the extra line. In the thin structure boundary, `dtang()` would correspond to $\nabla_{\mathbf{t}}$.

THERMAL CONDUCTIVITY TENSOR IN LOCAL BOUNDARY SYSTEMS

The thermal conductivity k describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T as in

$$\mathbf{q} = -k\nabla T$$

which is Fourier's law of heat conduction (see also The Physical Mechanisms under Heat Transfer).

The tensor components can be specified in the local coordinate system of the boundary, which is defined from the geometric tangent and normal vectors. The local x-direction, $\mathbf{e}^{x,\,\mathrm{loc}}$, is the surface tangent vector $\mathbf{t}_1$, and the local z-direction, $\mathbf{e}^{z,\,\mathrm{loc}}$, is the normal vector $\mathbf{n}$. Their cross product defines the third orthogonal direction such that:

$$\begin{aligned}\mathbf{e}^{x,\,\mathrm{loc}} &= \mathbf{t}_1\\ \mathbf{e}^{y,\,\mathrm{loc}} &= \mathbf{n}\times\mathbf{t}_1\\ \mathbf{e}^{z,\,\mathrm{loc}} &= \mathbf{n}\end{aligned}$$

From this, a transformation matrix between the local coordinate system and the global coordinate system can be constructed in the following way:

$$A = \begin{bmatrix} e_x^{x,\,\mathrm{loc}} & e_x^{y,\,\mathrm{loc}} & e_x^{z,\,\mathrm{loc}} \\ e_y^{x,\,\mathrm{loc}} & e_y^{y,\,\mathrm{loc}} & e_y^{z,\,\mathrm{loc}} \\ e_z^{x,\,\mathrm{loc}} & e_z^{y,\,\mathrm{loc}} & e_z^{z,\,\mathrm{loc}} \end{bmatrix}$$

The thermal conductivity tensor in the local coordinate system, k_{bnd}, is then expressed as

$$k_{\mathrm{bnd}} = AkA^{\mathrm{T}}$$

Theoretical Background of the Different Formulations

Three formulations are available for the modeling of heat transfer in thin structures defined as boundaries:

- The *general formulation*, using the Extra Dimension tool to solve the equations into the boundaries and through the thin structure's thickness
- The *thermally thin approximation*, a lumped formulation assuming that heat transfer mainly follows the tangential direction of the thin structure
- The *thermally thick approximation*, a lumped formulation assuming that heat transfer is dominant in the direction normal to the thin structure

They all derive from the energy equation established in Equation 4-13, and recalled here below:

$$\rho\frac{\partial E}{\partial t} + \rho\mathbf{u}\cdot\nabla E + \nabla\cdot(\mathbf{q}+\mathbf{q}_{\mathrm{r}}) = -(\sigma{:}\mathbf{D}) + Q$$

where E is the variable for the internal energy.

GENERAL FORMULATION

The general formulation uses the Extra Dimension tool to solve the equations through the thin structure's thickness. The thin structure has its domain represented by the product space between the lumped boundary and the additional dimension for the thickness. Applying the split of the gradient operator given earlier at Equation 4-33, the energy equation becomes

$$\rho\frac{\partial E_{\mathrm{s}}}{\partial t} + \rho\mathbf{u}\cdot(\nabla_{\mathbf{t}}E_{\mathrm{s}} + \nabla_{\mathbf{n}}E_{\mathrm{s}}) + \nabla\cdot(\mathbf{q}+\mathbf{q}_{\mathrm{r}}) = -(\sigma{:}\mathbf{D}) + Q \tag{4-34}$$

The $\nabla_{\mathbf{t}}$ operator is the tangential derivative in the thin structure boundary, and the $\nabla_{\mathbf{n}}$ operator is the derivation operator along the extra dimension which is normal to the thin structure (see Tangential and Normal Gradients). The subscript s appended on E (and T in the following) is here to recall that this variable lives in the product space of the thin structure.

Equation 4-34 comes along with Fourier's law of conduction:

$$\mathbf{q} = -k(\nabla_{\mathbf{t}}T_{\mathrm{s}} + \nabla_{\mathbf{n}}T_{\mathrm{s}}) \tag{4-35}$$

and constraints on the temperature at the extremities of the extra dimension:

$$T_{\mathrm{u}} = (T_{\mathrm{s}})_{L=0} \qquad T_{\mathrm{d}} = (T_{\mathrm{s}})_{L=d_s} \tag{4-36}$$

Here, d_{s} is the length of the extra dimension, or equivalently the thickness of the thin structure, and T_{u} and T_{d} are the temperature at the upside and the downside of the thin structure.

THERMALLY THIN APPROXIMATION

This formulation applies to a thin structure where heat transfer mainly follows the tangential direction. The gradient operator is then simplified to

$$\nabla T = \nabla_{\mathbf{t}} T$$

This assumption is often valid for thin structures that are good thermal conductors compared to the adjacent domains, and/or with fast convection along the tangential direction.

With these assumptions, Equation 4-13 becomes:

$$d_{\mathrm{s}}\rho\frac{\partial E}{\partial t} + d_{\mathrm{s}}\rho\mathbf{u}\cdot\nabla_{\mathbf{t}}E + \nabla_{\mathbf{t}}\cdot(\mathbf{q}_{\mathrm{s}} + \mathbf{q}_{\mathrm{r}}) = -d_{\mathrm{s}}(\sigma:\mathbf{D}) + d_{\mathrm{s}}Q + q_0 \tag{4-37}$$

$$\mathbf{q}_{\mathrm{s}} = -d_{\mathrm{s}}k\nabla_{\mathbf{t}}T \tag{4-38}$$

where d_{s} is the layer thickness (SI unit: m). The heat source Q is a density distributed in the layer while q_0 is the received out-of-plane heat flux.

In 2D, Equation 4-37 and Equation 4-38 have an additional factor, d_z, to account for the out-of-plane thickness.

When Equation 4-37 is solved in a boundary adjacent to a domain modeling heat transfer, the two entities exchange a certain amount of heat flux according to:

$$q_0 = \mathbf{n}\cdot\mathbf{q}$$

In this coupling relation, the outgoing heat flux $\mathbf{n}\cdot\mathbf{q}$ leaves the domain and is received in the source term q_0 by the adjacent thin layer modeled as a boundary. From the point of view of the domain, and neglecting thermoelastic effects, the following heat source is received from the thin structure:

$$-\mathbf{n}\cdot\mathbf{q} = d_{\mathrm{s}}Q_{\mathrm{s}} - d_{\mathrm{s}}\rho\frac{\partial E}{\partial t} - (d_{\mathrm{s}}\rho\mathbf{u}\cdot\nabla_{\mathbf{t}}E) - \nabla_{\mathbf{t}}\cdot(\mathbf{q}_{\mathrm{s}}+\mathbf{q}_{\mathrm{r}}) \tag{4-39}$$

Equations for all supported types of medium are presented in the next sections, Thin Layer, Thin Film, Fracture, and Thin Rod.

THERMALLY THICK APPROXIMATION

This formulation applies to a thin structure where heat transfer mainly follows the normal direction. The gradient operator is then simplified to

$$\nabla T = \nabla_{\mathbf{n}}T$$

This assumption is often valid for thin structures that are thermally resistive compared to the adjacent domains.

With these assumptions, Equation 4-13 becomes:

$$d_{\mathrm{s}}\rho\frac{\partial E}{\partial t} + d_{\mathrm{s}}\rho\mathbf{u}\cdot\nabla_{\mathbf{n}}E + \nabla_{\mathbf{n}}\cdot(\mathbf{q}_{\mathrm{s}}+\mathbf{q}_{\mathrm{r}}) = -d_{\mathrm{s}}(\sigma:\mathbf{D}) + d_{\mathrm{s}}Q + q_0 \tag{4-40}$$

$$\mathbf{q}_{\mathrm{s}} = -d_{\mathrm{s}}k\nabla_{\mathbf{n}}T \tag{4-41}$$

where d_{s} is the layer thickness (SI unit: m). The heat source Q is a density distributed in the layer while q_0 is the received out-of-plane heat flux.

In 2D, Equation 4-40 and Equation 4-41 have an additional factor, d_z, to account for the out-of-plane thickness.

When Equation 4-40 is solved in a boundary adjacent to a domain modeling heat transfer, the two entities exchange a certain amount of heat flux according to:

$$q_0 = \mathbf{n}\cdot\mathbf{q}$$

In this coupling relation, the outgoing heat flux $\mathbf{n}\cdot\mathbf{q}$ leaves the domain and is received in the source term q_0 by the adjacent thin layer modeled as a boundary. From the point of view of the domain, and neglecting thermoelastic effects, the following heat source is received from the thin structure:

$$-\mathbf{n}\cdot\mathbf{q} = d_{\mathrm{s}}Q_{\mathrm{s}} - d_{\mathrm{s}}\rho\frac{\partial E}{\partial t} - (d_{\mathrm{s}}\rho\mathbf{u}\cdot\nabla_{\mathbf{n}}E) - \nabla_{\mathbf{n}}\cdot(\mathbf{q}_{\mathrm{s}}+\mathbf{q}_{\mathrm{r}}) \tag{4-42}$$

To evaluate the normal gradient operation, $\nabla_{\mathbf{n}}$, temperatures T_{u} and T_{d} are introduced for the upside and downside of the thin structure boundary. They are defined from the heat flux across the thin resistive structure. At the middle of the thickness, the temperature, $T_{1/2}$, is approximated by $(1/2)(T_{\mathrm{u}}+T_{\mathrm{d}})$. The term $\nabla_{\mathbf{n}}\cdot(-k\nabla_{\mathbf{n}}T)$ is then given by:

$$\nabla_{\mathbf{n}}\cdot(-k\nabla_{\mathbf{n}}T)\approx -k_{\mathrm{s}}\frac{T_{\mathrm{d}}-2T_{1/2}+T_{\mathrm{u}}}{d_{\mathrm{s}}^2}$$

which can be seen as the sum of two contributive sources on the upside and on the downside of the boundary that compensate:

$$-k_{\mathrm{s}}\frac{T_{\mathrm{u}}-T_{\mathrm{d}}}{d_{\mathrm{s}}^2}\qquad -k_{\mathrm{s}}\frac{T_{\mathrm{d}}-T_{\mathrm{u}}}{d_{\mathrm{s}}^2}$$

leading to:

$$\rho C_p\frac{\partial T}{\partial t}+\left(-k_{\mathrm{s}}\frac{T_{\mathrm{u}}-T_{\mathrm{d}}}{d_{\mathrm{s}}^2}\right)+\left(-k_{\mathrm{s}}\frac{T_{\mathrm{d}}-T_{\mathrm{u}}}{d_{\mathrm{s}}^2}\right)+\nabla_{\mathbf{n}}\cdot\mathbf{q}_{\mathrm{r}} = Q$$

Ther term $\nabla_{\mathbf{n}}T$ is simply approximated by:

$$\nabla_{\mathbf{n}}T\approx\frac{T_{\mathrm{d}}-T_{\mathrm{u}}}{d_{\mathrm{s}}}$$

Equations for all supported types of medium are presented in the next sections, Thin Layer, Thin Film, Fracture, and Thin Rod.

UPSIDE, DOWNSIDE, AND EXTERIOR TEMPERATURES

This formulation is provided by the **Thermally thick approximation** option of the Thin Layer feature. Figure 4-3 shows how Thin Layer splits the temperature into T_u and T_d on interior boundaries:

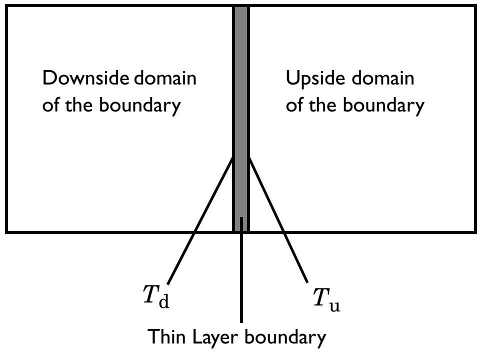


Figure 4-3: Upside and downside temperatures at a thin layer applied on an interior boundary. The thin layer is represented by the gray domain.

On exterior boundaries, it introduces a new degree of freedom represented by the variable T_extFace. Depending on whether the heat domain is on the upside or the downside of the boundary, T_extFace is equal to T_u or T_d and the same thing goes for the dependent variable T. An example is illustrated in the figure below:

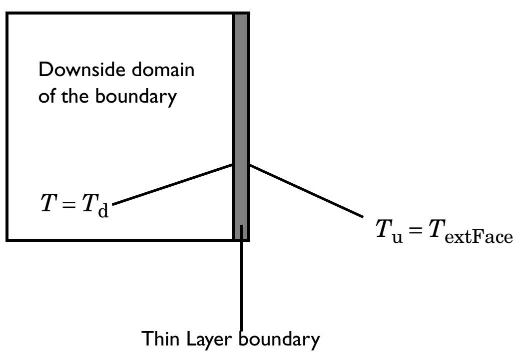


Figure 4-4: Upside and downside temperatures at a thin layer applied on an exterior boundary.

When using the pair Thin Layer node, the u and d subscripts refer to the upside and the downside of the pair, respectively, instead of the layer.

Like any pair feature, the Thin Layer condition contributes to any other pair feature. However, do not use two conditions of the same type on the same pair. In order to model a thin resistive layer made of several materials, use the **Multiple layers** option, which is available with the Heat Transfer Module.

FORMULATIONS AVAILABLE WITHIN THE FEATURES

Table 4-3 summarizes the formulations available within the thin structure features of the Heat transfer (`ht`) and Heat Transfer in Thin Shells (`htsh`) interfaces.

TABLE 4-3: FORMULATIONS AVAILABLE WITH THE THIN STRUCTURES FEATURES

FEATURE	GENERAL	THERMALLY THIN APPROXIMATION	THERMALLY THICK APPROXIMATION
Thin Layer (ht)	Yes	Yes	Yes
Thin Conductive Layer (htsh)	No	Yes	No
Thin Layered Shell (htsh)	Yes	No	No
Thin Film (ht, htsh)	Yes	Yes	No
Fracture (ht, htsh)	No	Yes	No

Thin Layer

Thin layers of solid materials can be considered as boundaries when their thickness is significantly smaller than the typical lengths of the adjacent domains.

GENERAL FORMULATION

With this formulation, multiple sandwiched layers with different material properties and thicknesses can be modeled. An additional 1D segmented line represents the multiple layers in the thin structure. In this extra dimension, the governing equation is derived from Equation 4-34 to give:

$$\rho_{si} C_{p,\,si} \frac{\partial T_{\mathrm{s}}}{\partial t} + \nabla_{\mathbf{t}} \cdot \mathbf{q}_{si} = Q_{si} \qquad (4\text{-}43)$$

$$\mathbf{q}_{si} = -k_{si}(\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \tag{4-44}$$

$$T_u = (T_s)_{L=0}$$

$$T_d = (T_s)_{L=d_s}$$

where T_s is an auxiliary dependent variable defined on the product space. The remaining quantities are recalled below:

- ρ_{si} is the density of layer i (SI unit: kg/m^3)
- $C_{p,\,si}$ is the heat capacity of layer i (SI unit: J/(kg·K))
- k_{si} is the thermal conductivity of layer i (SI unit: W/(m·K))
- Q_{si} is the heat source applied to layer i (SI unit: W/m^3)
- d_s is the shell thickness (SI unit: m)

The constraint $T = T_s$ is specified on each side of the extra dimension to connect T to T_s.

See Thin Layer with **Layer type** set as **General** or more information about the boundary feature solving Equation 4-43 and Equation 4-44.

THERMALLY THIN APPROXIMATION

The Heat Transfer Module supports heat transfer in thermally thin structures in 3D, 2D, and 2D axisymmetry. The material in the thin structure might be a good thermal conductor for this approximation to be valid. For example, in a printed circuit with copper traces, where the traces are often good thermal conductors compared to the board's substrate material.

The thermally thin approximation is derived from Equation 4-37 to Equation 4-39. Inside the thin layer, the heat equation becomes:

$$d_s \rho C_{p,\,s} \frac{\partial T}{\partial t} + \nabla_{\mathbf{t}} \cdot \mathbf{q}_s = d_s Q_s + q_0 \tag{4-45}$$

$$\mathbf{q}_s = -d_s k \nabla_{\mathbf{t}} T \tag{4-46}$$

where d_s is the layer thickness (SI unit: m). The heat source Q_s is a density distributed in the layer while q_0 is the received out-of-plane heat flux.

In 2D, Equation 4-45 and Equation 4-46 have an additional factor, d_z, to account for the out-of-plane thickness.

From the point of view of the domain, the following heat source, derived from Equation 4-39, is received from the layer:

$$-\mathbf{n} \cdot \mathbf{q} = d_s Q_s - d_s \rho C_{p,s} \frac{\partial T}{\partial t} - \nabla_{\mathbf{t}} \cdot \mathbf{q}_s \tag{4-47}$$

See Thin Layer with **Layer type** set as **Thermally thin approximation** for more information about the boundary feature solving Equation 4-47. See The Heat Transfer in Thin Shells Interface for more information about the physics interface solving Equation 4-45.

- *Heat Transfer in a Surface-Mount Package for a Silicon Chip*: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/surface_mount_package**
- *Silica Glass Block Coated with a Copper Layer*: Application Library path **Heat_Transfer_Module/Tutorials,_Thin_Structure/copper_layer**

THERMALLY THICK APPROXIMATION

When a thin layer is formed of one or more thermally resistive materials, it can be defined through its thermal resistance:

$$R_s = \frac{d_s}{k_s}$$

Neglecting time-dependent terms, the heat flux across the thermally thick structure is derived from Equation 4-42 and gives

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -k_s \frac{T_u - T_d}{d_s} + \frac{1}{2} d_s Q_s \tag{4-48}$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = -k_s \frac{T_d - T_u}{d_s} + \frac{1}{2} d_s Q_s \tag{4-49}$$

where the u and d subscripts refer to the upside and downside of the layer, respectively.

When the material has a multilayer structure k_s and d_s in the expressions above are replaced by d_{tot} and k_{tot}, which are defined according to Equation 4-50 and Equation 4-51:

$$d_{tot} = \sum_{j=1}^{n} d_{sj} \tag{4-50}$$

$$k_{tot} = \frac{d_{tot}}{\sum_{j=1}^{n} \frac{d_{sj}}{k_{sj}}} \tag{4-51}$$

where n is the number of layers.

See Thin Layer with **Layer type** set as **Thermally thick approximation** for more information about the boundary feature solving Equation 4-48 and Equation 4-49.

Thin Film

Thin films of fluid can be considered as boundaries of thickness significantly smaller than the typical lengths of the overall model.

GENERAL FORMULATION

With this formulation, heat transfer is modeled in the whole film, including its thickness. An additional 1D segmented line represents the thickness in the thin film. In this extra dimension, the governing equation is derived from Equation 4-34 to give:

$$\rho C_p \frac{\partial T_s}{\partial t} + \rho C_p \mathbf{u} \cdot (\nabla_\mathbf{t} T_s + \nabla_\mathbf{n} T_s) + \nabla_\mathbf{t} \cdot \mathbf{q}_f = Q_f \tag{4-52}$$

$$\mathbf{q}_f = -k(\nabla_\mathbf{t} T_s + \nabla_\mathbf{n} T_s) \tag{4-53}$$

$$T_u = (T_s)_{L=0}$$

$$T_{\mathrm{d}} = (T_{\mathrm{s}})_{L = d_f}$$

where T_{s} is an auxiliary dependent variable defined on the product space. The remaining quantities are recalled below:

- ρ is the density (SI unit: $\mathrm{kg/m^3}$)
- C_p is the heat capacity (SI unit: $\mathrm{J/(kg{\cdot}K)}$)
- k is the thermal conductivity (SI unit: $\mathrm{W/(m{\cdot}K)}$)
- Q_{f} is the heat source applied to the film (SI unit: $\mathrm{W/m^3}$)
- d_{f} is the film thickness (SI unit: m)

The constraint $T = T_{\mathrm{s}}$ is specified on each side of the extra dimension to connect T to T_{s}.

See Thin Film with **Thin film model** set as **General** for more information about the boundary feature solving Equation 4-52 and Equation 4-53.

THERMALLY THIN APPROXIMATION

The thermally thin approximation is derived from Equation 4-37 to Equation 4-39. Inside the thin layer, the heat equation becomes:

$$d_{\mathrm{f}}\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla_{\mathbf{t}}T\right) + \nabla_{\mathbf{t}}\cdot\mathbf{q}_{\mathrm{f}} = d_{\mathrm{f}}Q_{\mathrm{f}} + q_0 \tag{4-54}$$

$$\mathbf{q}_{\mathrm{f}} = -d_{\mathrm{f}}k\nabla_{\mathbf{t}}T \tag{4-55}$$

where d_{f} is the film thickness (SI unit: m). The heat source Q_{f} is a density distributed in the layer while q_0 is the received out-of-plane heat flux.

In 2D, Equation 4-45 and Equation 4-46 have an additional factor, d_z, to account for the out-of-plane thickness.

From the point of view of the domain, the following heat source, derived from Equation 4-39, is received from the layer:

$$-\mathbf{n}\cdot\mathbf{q} = d_\mathrm{f}Q_\mathrm{f} - d_\mathrm{f}\rho C_p\frac{\partial T}{\partial t} - d_\mathrm{f}\rho C_p\mathbf{u}\cdot\nabla_\mathbf{t}T - \nabla_\mathbf{t}\cdot\mathbf{q}_\mathrm{f} \tag{4-56}$$

See Thin Film with **Thin film model** set as **Thermally thin approximation** for more information about the boundary feature solving Equation 4-54. See The Heat Transfer in Thin Films Interface for more information about the physics interface solving Equation 4-56.

Fracture

When fractures occur in porous media, fluid flow tends to move faster than in the bulk medium. The transport of heat occurs faster in the fractures that in the surrounding medium, so in this sense, heat transfer in fractures filled with fluids is more similar to a highly conductive layer than to a thin thermally resistive layer.

The mass transport in fractures can be modeled as Darcy's law in a thin sheet of porous medium:

$$\mathbf{u} = \frac{\kappa}{\mu}\nabla_t p$$

where $\mathbf{u}$ is the tangential Darcy's velocity (SI unit: m/s), κ is the fracture permeability (SI unit: m^2), μ the fluid's dynamic viscosity (SI unit: Pa·s), and $\nabla_t p$ the tangential gradient of the fluid's pressure.

Typically, Darcy's Law with tangential derivatives is solved to compute mass transport, so in addition to the fluid properties, the fracture should define its own permeability (or hydraulic conductivity in case the fluid is water), porosity, and fracture thickness.

For heat transfer in fractures, the fracture also needs to define the density of the porous sheet, heat capacity, and thermal conductivity. The effective thermal conductivity of the fracture must be adjusted to the fracture porosity and thermal conductivity of the fluid. In rocks and geological formations, the fracture might also contain highly conductive material, different than the bulk porous matrix.

The equation to solve for computing heat transfer in fractures is derived from Equation 4-37 to Equation 4-39 and using the procedure detailed in Theory for Heat Transfer in Porous Media to apply the mixture rule on solid and fluid internal energies. The resulting equations are:

$$d_{\mathrm{fr}}(\rho C_p)_{\mathrm{eff}}\frac{\partial T}{\partial t}+d_{\mathrm{fr}}\rho C_p\mathbf{u}\cdot\nabla_{\mathbf{t}}T+\nabla_{\mathbf{t}}\cdot\mathbf{q}_{\mathrm{fr}} = d_{\mathrm{fr}}Q+q_0 \tag{4-57}$$

$$\mathbf{q}_{\mathrm{fr}} = -d_{\mathrm{fr}}k_{\mathrm{eff}}\nabla_{\mathbf{t}}T \tag{4-58}$$

Here $(\rho C_p)_{\mathrm{eff}}$ is the effective heat capacity at constant pressure of the fracture-fluid volume, ρ is the fluid's density, C_p is the fluid's heat capacity at constant pressure, $\mathbf{q}_{\mathrm{fr}}$ is the conductive heat flux in the fracture-fluid volume, k_{eff} is the effective thermal conductivity of the fluid-fracture mixture, and Q is a possible heat source.

From the point of view of the domain, the following heat source, derived from Equation 4-39, is received from the fracture:

$$-\mathbf{n}\cdot\mathbf{q} = d_{\mathrm{fr}}Q_0-d_{\mathrm{fr}}(\rho C_p)_{\mathrm{eff}}\frac{\partial T}{\partial t}-d_{\mathrm{fr}}\rho C_p\mathbf{u}\cdot\nabla_{\mathbf{t}}T-\nabla_{\mathbf{t}}\cdot\mathbf{q}_{\mathrm{fr}} \tag{4-59}$$

See Fracture for more information about the boundary feature solving Equation 4-59. See The Heat Transfer in Fractures Interface for more information about the physics interface solving Equation 4-57.

Thin Rod

The Thin Rod feature is similar to Thin Layer with **Layer type** set as **Thermally thin approximation**. It provides a lumped heat transfer model to model thermally thin rods as edges.

The edge condition reads:

$$\int_{S(R)} Qds = A_1Q_1-A_1\rho_1C_{p,1}+\frac{\partial T}{\partial t}-\nabla_{\mathbf{t}}\cdot\mathbf{q}_1 \tag{4-60}$$

$$\mathbf{q}_1 = -A_1k_1\nabla_{\mathbf{t}}T \tag{4-61}$$

with

$$A_1 = \pi r_1^{\ 2}$$

See Thin Rod for node information.

Theory for Surface-to-Surface Radiation

In addition to conduction and convection, the third mechanism for heat transfer is radiation. Consider an environment with fully transparent or fully opaque objects. Thermal radiation denotes the stream of electromagnetic waves emitted from a body at a certain temperature.

The Surface-To-Surface Radiation Interface theory is described in this section:

- Deriving the Radiative Heat Flux
- Wavelength Dependence of Surface Emissivity and Absorptivity
- The Radiosity Method for Diffuse-Gray Surfaces
- The Radiosity Method for Diffuse-Spectral Surfaces
- View Factor Evaluation

Deriving the Radiative Heat Flux

In Figure 4-5, consider a point P located on a surface that has an emissivity ε, reflectivity ρ, absorptivity α, refractive index n, and temperature T. The body is assumed opaque, which means that no radiation is transmitted through the body. This is true for most solid bodies.

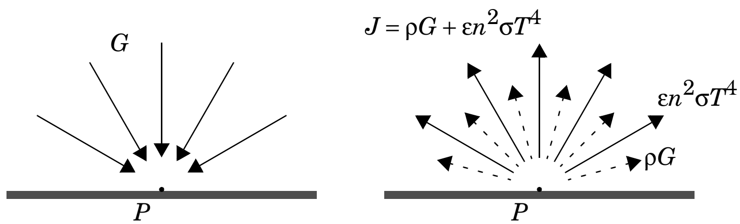


Figure 4-5: Incoming irradiation (left), outgoing radiosity (right).

The total incoming radiative flux at P is called *irradiation* and denoted G. The total outgoing radiative flux at P is called *radiosity* and denoted J. This radiosity is the sum of reflected and emitted radiation:

$$J = \rho G + \varepsilon e_b(T) \tag{4-62}$$

According to the Stefan-Boltzmann law, $e_b(T)$ is the power radiated across all wavelengths and depends on the forth power of the temperature:

$$e_b(T) = n^2\sigma T^4$$

The net inward radiative heat flux, q, is then given by the difference between the irradiation and the radiosity:

$$q = G - J \tag{4-63}$$

Using Equation 4-62 and Equation 4-63, J can be eliminated and a general expression is obtained for the net inward heat flux into the opaque body based on G and T.

$$q = (1-\rho)G - \varepsilon e_b(T) \tag{4-64}$$

Most opaque bodies also behave as ideal gray bodies, meaning that the absorptivity and emissivity are equal, and the reflectivity is therefore obtained from the following relation:

$$\alpha = \varepsilon = 1 - \rho \tag{4-65}$$

Thus, for ideal gray bodies, q is given by:

$$q = \varepsilon(G - e_b(T)) \tag{4-66}$$

This is the expression used for the radiative boundary condition.

Wavelength Dependence of Surface Emissivity and Absorptivity

The surface properties for radiation, the emissivity, and absorptivity can be dependent on the angle of emission or absorption, the surface temperature, or the radiation wavelength. The emissivity and absorptivity are defined in Ref. 15.

The Surface-to-Surface Radiation interface in the Heat Transfer module implements the radiosity method that enables arbitrary temperature dependence and assumes that the emissivity and absorptivity is independent of the angle of emission and absorption. It is also possible to account for wavelength dependence on the surface emissivity and absorptivity.

PLANCK SPECTRAL DISTRIBUTION

The Planck's distribution of emissive power for a blackbody in vacuum is given as a function of surface temperature and wavelength.

The blackbody hemispherical emissive power (SI unit: W/(m^3·sr)), is denoted $e_{b,\lambda}(\lambda, T)$, and defined as (1-37 in Ref. 15):

$$e_{b,\lambda}(\lambda, T) = \frac{2\pi n^2 C_1}{\lambda^5\left(e^{\frac{C_2}{\lambda T}} - 1\right)} \tag{4-67}$$

where:

- the two constants C_1 (SI unit: W·m^2/sr) and C_2 (SI unit: m·K) are given by

$$C_1 = hc_0^2 \qquad C_2 = \frac{hc_0}{k_B}$$

- h is the Planck constant (SI unit: J·s)
- k_B is the Boltzmann constant (SI unit: J/K)
- c_0 is the speed of the light in vacuum (SI unit: m/s)
- λ is the wavelength in vacuum (SI unit: m)
- n is the refractive index of the media (SI unit: 1), equal to 1 in vacuum

Figure 4-6 and Figure 4-7 show the hemispherical spectral emissive power for a blackbody at 5780 K (the Sun's blackbody temperature) and for a blackbody at 300 K. The dotted vertical lines delimit the visible spectrum (from 0.4 µm to 0.7 µm).

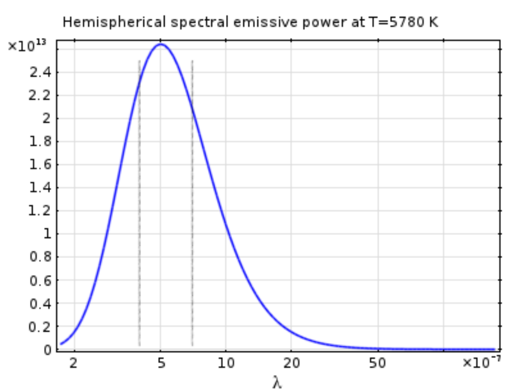


Figure 4-6: Planck distribution of a blackbody at 5780 K.

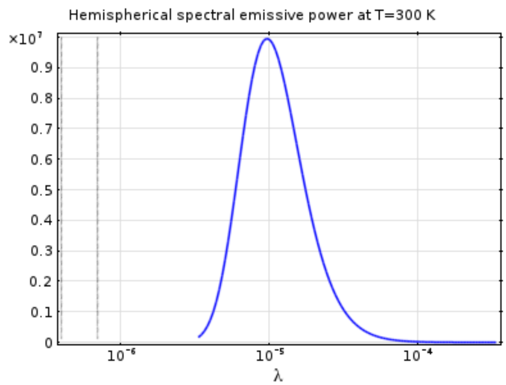


Figure 4-7: Planck distribution of a blackbody at 300 K.

The integral of $e_{\mathrm{b},\lambda}(\lambda, T)$ over a spectral band represents the power radiated on the spectral band and is defined by

$$\int_{\lambda_1}^{\lambda_2} e_{\mathrm{b},\lambda}(\lambda, T)d\lambda = F_{\lambda_1 T \to \lambda_2 T}\int_0^{\infty} e_{\mathrm{b},\lambda}(\lambda, T)d\lambda$$

where $F_{\lambda_1 T \to \lambda_2 T}$ is the fractional blackbody emissive power,

$$F_{\lambda_1 T \to \lambda_2 T} = \frac{\int_{\lambda_1}^{\lambda_2} e_{\mathrm{b},\lambda}(\lambda, T)d\lambda}{\int_0^{\infty} e_{\mathrm{b},\lambda}(\lambda, T)d\lambda}$$

Recall the Stefan-Boltzmann law that computes the power radiated across all wavelengths:

$$\int_0^{\infty} e_{\mathrm{b},\lambda}(\lambda, T)d\lambda = e_{\mathrm{b}}(T) = n^2\sigma T^4$$

where n is the refractive index, and σ is the Stefan-Boltzmann constant equal to $5.67 \cdot 10^{-8}$ W/(m^2·K^4). The power radiated in the spectral band $[\lambda_1, \lambda_2]$ becomes:

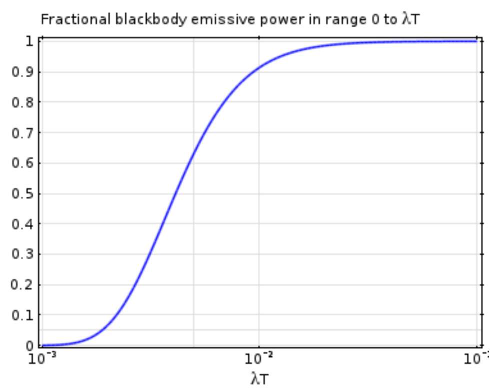
$$\int_{\lambda_1}^{\lambda_2} e_{b,\lambda}(\lambda, T)d\lambda = F_{\lambda_1 T \to \lambda_2 T} e_b(T)$$

The function $e_b(T)$ is available as a predefined function via `ht.feb(T)` in the Heat Transfer interfaces.

Notice that:

$$F_{\lambda_1 T \to \lambda_2 T} = F_{0 \to \lambda_2 T} - F_{0 \to \lambda_1 T} \text{ and } F_{0 \to \infty} = 1$$

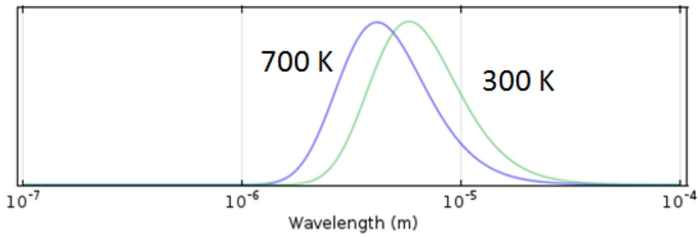
The figure below shows the value of $F_{0 \to \lambda T}$ for different values of λT.



DIFFUSE-GRAY SURFACES

Diffuse-gray surfaces correspond to the hypothesis that surface properties are independent of the radiation wavelength and angle between the surface normal and the radiation direction.

The assumption that the surface emissivity is independent of the radiation wavelength is often valid when most of the radiative power is concentrated on a relatively narrow spectral band. This is likely the case when the radiation is emitted by a surface at temperatures in limited range.



This setting is rarely applicable if there is solar radiation.

SOLAR AND AMBIENT SPECTRAL BANDS

When solar radiation is part of the model, it is possible to enhance a diffuse-gray surface model by considering two spectral bands: one for short wavelengths and one for large wavelengths.

It is interesting to notice that about 97% of the radiated power from a blackbody at 5800 K is at wavelengths of 2.5 µm or shorter, and 97% of the radiated power from a blackbody at 700 K is at wavelengths of 2.5 µm or longer (see Figure 4-8).

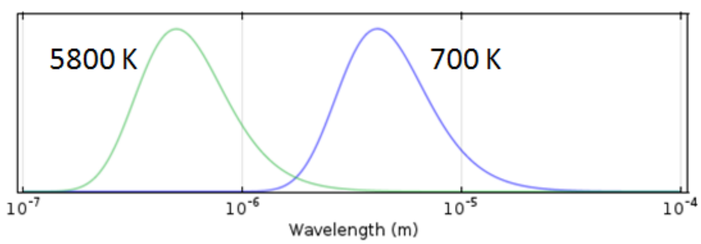


Figure 4-8: Normalized Planck distribution of blackbodies at 700 K and 5800 K.

Many problems have a solar load, but the peak temperatures are below 700 K.

In such cases, it is appropriate to use a two-band approach with

- A solar band for wavelengths shorter than 2.5 µm
- An ambient band for wavelengths above 2.5 µm

For each surface, properties are then described in terms of a solar absorptivity and an emissivity.

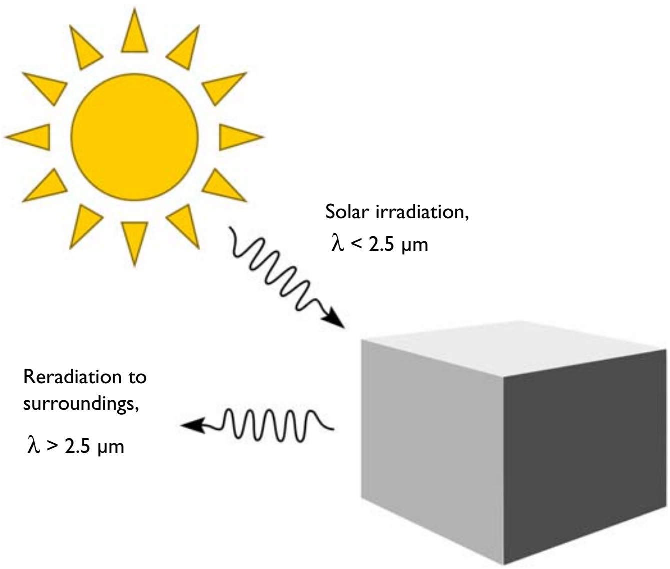


Figure 4-9: Absorption of solar radiation and emission to the surroundings.

By splitting the bands at the default of 2.5 µm, the fraction of absorbed solar radiation on each surface is defined primarily by the solar absorptivity.

The reradiation at longer wavelengths (objects below ~700 K) and the reabsorption of this radiation is defined primarily via the emissivity

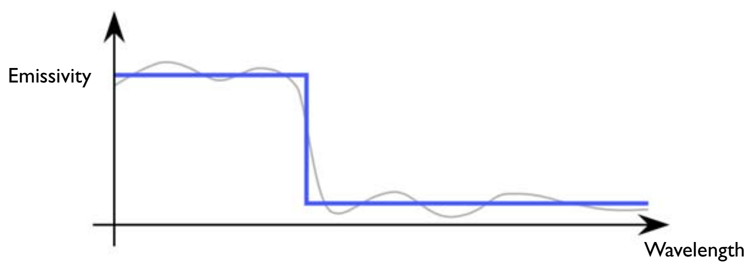
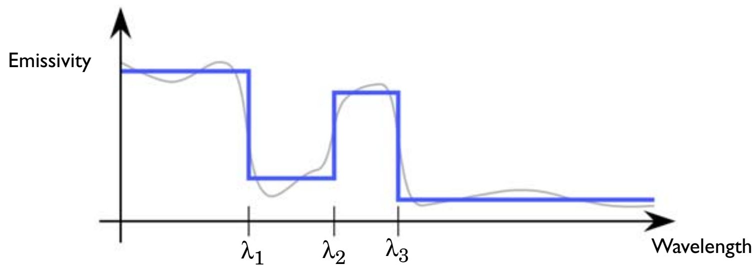


Figure 4-10: Solar and ambient spectral band approximation of the surface emissivity by a constant per band emissivity.

GENERAL DIFFUSE-SPECTRAL SURFACES

Diffuse-spectral surfaces correspond to the hypothesis that surface properties are wavelength dependent but independent of the angle between the surface normal and the radiation direction.

The heat transfer module enables to define constant surface properties per spectral bands (with up to 5 spectral bands) and to adjust spectral intervals endpoints.



The multiple spectral bands approach is used in cases when the surface emissivity varies significantly over the bands of interest.

The Radiosity Method for Diffuse-Gray Surfaces

The heat transfer by radiation is combined with convective and conductive heat transfer through a source term added to the heat equation along with the other contributions from the heat flux and boundary heat source boundary conditions. Recalling Equation 4-63, this source account for the difference between incident radiation, or ***irradiation***, G, and radiation leaving the surface, or ***radiosity***, J:

$$q = G - J$$

The radiosity, J, is given in Equation 4-62. It is the sum of reflected and emitted radiation. For diffuse-gray surfaces, J is defined by:

$$J = (1-\varepsilon)G + \varepsilon e_b(T)$$

Here

- G is the incoming radiative heat flux, or *irradiation* (SI unit: W/m^2)
- ε is the surface emissivity (SI unit: 1), a dimensionless number in the range $0 \leq \varepsilon \leq 1$. The diffuse-gray surface hypothesis corresponds to surfaces where ε is independent of the radiation wavelength.

- $e_b(T)$ is the blackbody hemispherical total emissive power (SI unit: W/m^2).
- T is the surface temperature (SI unit: K).

The irradiation, G, at a given point is split into three contributions according to:

$$G = G_m + G_{ext} + G_{amb} \tag{4-68}$$

where:

- G_m is the mutual irradiation, coming from other boundaries in the model (SI unit: W/m^2).
- G_{ext} is the irradiation from external radiation sources (SI unit: W/m^2). It is the sum of the products, for each external source, of the external heat sources view factor F_{ext} by the corresponding source radiosity:

$$G_{ext} = \sum F_{ext} P_s + \sum F_{ext} q_{0,s}$$

  The first term of the sum gathers radiation sources located on a point. The second term stands for directional radiative sources.
- G_{amb} is the ambient irradiation (SI unit: W/m^2), defined as:

$$G_{amb} = F_{amb} e_b(T_{amb})$$

- F_{amb} is an *ambient view factor;* its value is equal to the fraction of the field of view that is not covered by other boundaries. Therefore, by definition, $0 \le F_{amb} \le 1$ at all points.
- T_{amb} is the assumed far-away temperature (SI unit: K) in the directions included in F_{amb}.

The Surface-To-Surface Radiation Interface includes the following radiation types:

- Diffuse Surface is the default radiation type. It requires accurate evaluation of the mutual irradiation, G_m. The incident radiation at one point on the boundary is a function of the radiosity, J, at every other point in view. The radiosity, in turn, is a function of G_m, which leads to an implicit radiation balance:

$$J = (1-\varepsilon)(G_m(J) + G_{ext} + G_{amb}) + \varepsilon e_b(T) \tag{4-69}$$

- Diffuse Mirror is a variant of the Diffuse Surface radiation type with $\varepsilon = 0$. Reradiation surfaces are common as an approximation of a surface that is well insulated on one side and for which convection effects can be neglected on the opposite (radiating) side (see Ref. 16). It resembles a mirror that absorbs all irradiation and then radiates it back in all directions.

- Prescribed Radiosity makes it possible to specify *graybody radiation*. The radiosity expression is then $\varepsilon e_b(T)$. A user-defined surface radiosity expression can also be defined.

The Surface-to-Surface Radiation interface handles the radiosity J as a shape function unless J is prescribed.

The Radiosity Method for Diffuse-Spectral Surfaces

For a general diffuse-spectral surface:

$$J = \int_0^{\infty}((1-\varepsilon(\lambda, T))G(\lambda) + \varepsilon(\lambda, T)e_{b,\lambda}(\lambda, T))d\lambda$$

where

- $\varepsilon(\lambda, T)$ is the hemispherical spectral surface emissivity, a dimensionless quantity in the range $0 \le \varepsilon \le 1$. Diffuse-spectral surface corresponds to a surface where ε is dependent on the radiation wavelength and surface temperature.
- T is the surface temperature (SI unit: K).
- $e_{b,\lambda}(\lambda, T)$ is the blackbody hemispherical emissive power (SI unit: $W/(m^3 \cdot sr)$) defined in Equation 4-67.

The Surface-To-Surface Radiation Interface assumes that the surface emissivity and opacity properties are constant per spectral band. It defines N spectral bands ($N = 2$ when solar and ambient radiation model is used),

$$\begin{cases} B_i = [\lambda_{i-1}, \lambda_i] & \text{for } 1 \le i \le N \\ \lambda_0 = 0 & \\ \lambda_N = \infty & \end{cases}$$

so that the radiosity has a custom definition in each interval:

$$J = \sum_{i=1}^{N} J_i$$

$$J_i = (1-\varepsilon_i)G_i + \varepsilon_i e_b(T)$$

The surface properties can then be defined per spectral band:

- Surface emissivity on B_i: $\varepsilon_i(T) = \varepsilon(\lambda, T)$ for λ in the interval B_i
- Ambient irradiation on B_i, assuming that the ambient fractional emissive power corresponds to the one of a blackbody at temperature T_{amb}:

$$G_{\mathrm{amb},\,i} = \int_{\lambda = \lambda_{i-1}}^{\lambda_i} G_{\mathrm{amb}}(\lambda)d\lambda = F_{\lambda_{i-1}T \to \lambda_i T} F_{\mathrm{amb}} e_{\mathrm{b}}(T_{\mathrm{amb}})$$

- External radiation sources on B_i with $q_{0,\,\mathrm{s},\,i}$ and $P_{\mathrm{s},\,i}$ the external radiation source heat flux and heat rate, respectively, over B_i:

$$G_{\mathrm{ext},\,i} = \int_{\lambda = \lambda_{i-1}}^{\lambda_i} G_{\mathrm{ext}}(\lambda)d\lambda = F_{\mathrm{ext},\,i}(i_{\mathrm{s}})q_{0,\,\mathrm{s},\,i}$$

  or

$$G_{\mathrm{ext},\,i} = \int_{\lambda = \lambda_{i-1}}^{\lambda_i} G_{\mathrm{ext}}(\lambda)d\lambda = F_{\mathrm{ext},\,i}(i_{\mathrm{s}})P_{\mathrm{s},\,i}$$

  When the external source fractional emissive power corresponds to the one of a blackbody at T_{ext}, external radiation sources on B_i can be defined from the external radiation source heat flux, $q_{0,\,\mathrm{s}}$, and heat rate, P_{s}, over all wavelengths:

$$G_{\mathrm{ext},\,i} = F_{\mathrm{ext},\,i} F_{\lambda_{i-1}T \to \lambda_i T}(i_{\mathrm{s}})q_{0,\,\mathrm{s}}$$

  or

$$G_{\mathrm{ext},\,i} = F_{\mathrm{ext},\,i} F_{\lambda_{i-1}T \to \lambda_i T}(i_{\mathrm{s}})P_{\mathrm{s}}$$

The Surface-To-Surface Radiation Interface includes the following radiation types:

- Diffuse Surface is the default radiation type. The incident radiation over the B_i spectral band at one point of the boundary is a function of the radiosity, J_i (SI unit: $\mathrm{W/m^2}$), at every other point in view. The radiosity, in turn, is a function of $G_{\mathrm{m},\,i}$, which leads to an implicit radiation balance:

$$J_i = (1-\varepsilon_i)(G_{\mathrm{m},\,i}(J_i) + G_{\mathrm{ext},\,i} + G_{\mathrm{amb},\,i}) + \varepsilon_i e_{\mathrm{b}}(T) \qquad (4\text{-}70)$$

- Diffuse Mirror is a variant of the Diffuse Surface radiation type with $\varepsilon_i = 0$. Reradiation surfaces are common as an approximation of a surface that is well insulated on one side and for which convection effects can be neglected on the opposite (radiating) side (see Ref. 16). It resembles a mirror that absorbs all irradiation and then radiates it back in all directions.

- Prescribed Radiosity makes it possible to specify the surface radiation for each spectral band. Using the graybody radiation definition, the radiosity is then $F_{\lambda_1 T \to \lambda_2 T} e_{\mathrm{b}}(T)$. A user-defined surface radiosity expression can also be defined.

The Surface-to-Surface Radiation interface handles the radiosity J_i as a shape function unless J_i is prescribed.

View Factor Evaluation

The strategy for evaluating *view factors* is central to any radiation simulation. Loosely speaking, a view factor is a measure of how much influence the radiosity at a given part of the boundary has on the irradiation at some other part.

The quantities G_{m} and F_{amb} in Equation 4-69 are not strictly view factors in the traditional sense. Instead, F_{amb} is the view factor of the ambient portion of the field of view, which is considered to be a single boundary with constant radiosity

$$J_{\mathrm{amb}} = e_{\mathrm{b}}(T_{\mathrm{amb}})$$

On the other hand, G_{m} is the integral over all visible points of a differential view factor, multiplied by the radiosity of the corresponding source point. In the discrete model, think of it as the product of a view factor matrix and a radiosity vector. This is, however, not necessarily the way the calculation is performed.

Consider a point P on a surface as in Figure 4-11. It can be seen by points on other surfaces such as S' in the figure, as well as the ambient surrounding, S_{amb}. Assume that the points on S' have a local radiosity, J', while the ambient surrounding has a constant temperature, T_{amb}.

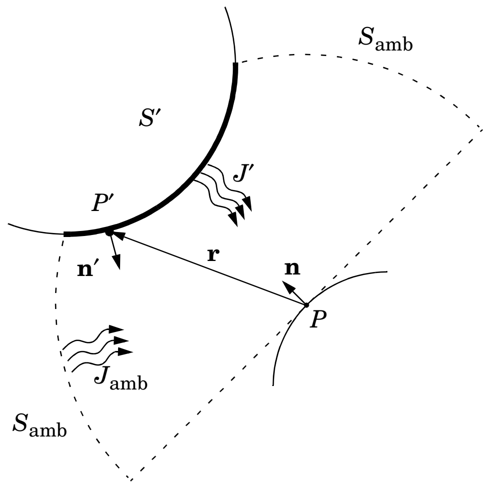


Figure 4-11: Example geometry for surface-to-surface radiation.

The mutual irradiation at point P is given by the following surface integral:

$$G_{\mathrm{m}} = \int_{S'} \frac{(-\mathbf{n}' \cdot \mathbf{r})(\mathbf{n} \cdot \mathbf{r})}{\pi|\mathbf{r}|^4} J' ds$$

The heat flux that arrives from P' depends on the local radiosity J' projected onto P. The projection is computed using the normal vectors $\mathbf{n}$ and $\mathbf{n}'$ along with the vector $\mathbf{r}$, which points from P to P'.

The ambient view factor, F_{amb}, is determined from the integral of the surrounding surfaces S', here denoted as F':

$$F_{\mathrm{amb}} = 1 - F' = 1 - \int_{S'} \frac{(-\mathbf{n}' \cdot \mathbf{r})(\mathbf{n} \cdot \mathbf{r})}{\pi|\mathbf{r}|^4} ds$$

The two last equations plug into Equation 4-68 to yield the final equation for irradiative flux.

The equations used so far apply to the general 3D case. 2D geometries result in simpler integrals. For the 2D case, the resulting equations for the mutual irradiation and ambient view factor are

$$G_{\mathrm{m}} = \int_{S_\perp'} \frac{(-\mathbf{n}' \cdot \mathbf{r}_\perp)(\mathbf{n} \cdot \mathbf{r}_\perp)}{2|\mathbf{r}_\perp|^3} J' ds \qquad (4\text{-}71)$$

$$F_{\mathrm{amb}} = 1 - \int_{S_\perp'} \frac{(-\mathbf{n}' \cdot \mathbf{r}_\perp)(\mathbf{n} \cdot \mathbf{r}_\perp)}{2|\mathbf{r}_\perp|^3} ds$$

where the integral over $S_\perp'$ denotes the line integral along the boundaries of the 2D geometry.

In axisymmetric geometries or when a symmetry plan is defined, the irradiation and ambient view factor cannot be computed directly from a closed-form expression. Instead, a virtual geometry must be constructed, and the view factors evaluated according to Equation 4-71.

A separate evaluation is performed for each unique point where G_{m} or F_{amb} is requested, typically for each quadrature point during solution. Differential view factors are normally computed only once, the first time they are needed, and then stored in memory until next time the model definition or the mesh is changed.

The Heat Transfer Module supports two surface-to-surface radiation methods, which are selected in the Radiation Settings section in a Heat Transfer interface:

- Hemicube
- Discrete area integration

View factors are always calculated directly from the mesh, which is a polygonal representation of the geometry. To improve the accuracy of the radiative heat transfer simulation, the mesh must be refined rather than raising the element order.

VIEW FACTOR FOR EXTERNAL RADIATION SOURCES

In 3D, the view factor for a point at finite distance is given by

$$\frac{\cos\theta}{4\pi r^2}$$

where θ is the angle between the normal to the irradiated surface and the direction of the source, and r is the distance from the source. For a source at infinity, the view factor is given by $\cos\theta$.

In 2D the view factor for a point at finite distance is given by

$$\frac{\cos\theta}{2\pi r}$$

and the view factor for a source at infinity is $\cos\theta$.

SOLAR POSITION

The Sun is the most common example of an external radiation source. The position of the Sun is necessary to determine the direction of the corresponding external radiation source. The direction of sunlight (zenith angle and the solar elevation) is automatically computed from the latitude, longitude, time zone, date, and time using similar a method as described in Ref. 15. The estimated solar position is accurate for a date between year 2000 and 2199, due to an approximation used in the Julian Day calendar calculation.

The zenith angle, θ_s, and azimuth angle, φ_s, of the Sun are converted into a direction vector $\mathbf{i}_s = (i_{sx}, i_{sy}, i_{sz})$ in Cartesian coordinates assuming that the north, the west, and the up directions correspond to the x, y, and z directions, respectively, in the model. The relation between θ_s, φ_s, and $\mathbf{i}_s$ is given by:

$$\begin{aligned} i_{sx} &= -\cos(\varphi_s)\sin(\theta_s) \\ i_{sy} &= \sin(\varphi_s)\sin(\theta_s) \\ i_{sz} &= -\cos(\theta_s) \end{aligned}$$

RADIATION IN AXISYMMETRIC GEOMETRIES

For an axisymmetric geometry, G_m and F_{amb} must be evaluated in a corresponding 3D geometry obtained by revolving the 2D boundaries around the axis. COMSOL Multiphysics creates this virtual 3D geometry by revolving the 2D boundary mesh into a 3D mesh. The resolution can be controlled in the azimuthal direction by setting the number of azimuthal sectors, which is the same as the number of elements to a full revolution. Try to balance this number against the mesh resolution in the *rz*-plane. This number, the azimuthal sectors, is accessible from the Radiation Settings section in physics interfaces for heat transfer.

Select between the hemicube and the direct area integration methods also in axial symmetry. Their settings work the same way as in 3D.

While G_m and F_amb are in fact evaluated in a full 3D, the number of points where they are requested is limited to the quadrature points on the boundary of a 2D geometry. The savings compared to a full 3D simulation are therefore substantial despite the full 3D view factor code being used.

Theory for Radiation in Participating Media

Radiation and Participating Media Interactions

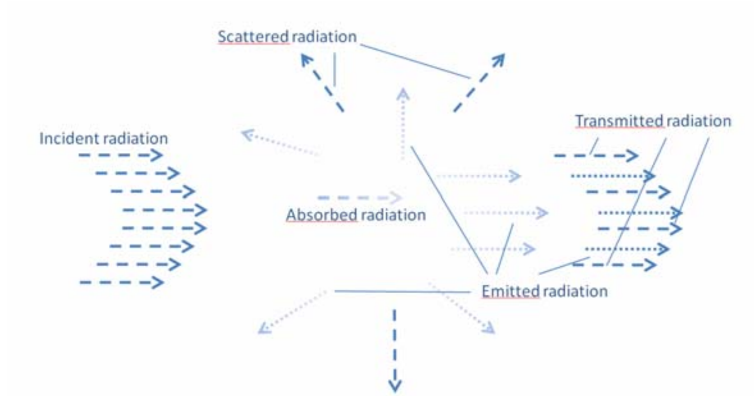


Figure 4-12: Example of interactions between participating media and radiation.

In some applications the medium is not completely transparent and the radiation rays interact with the medium.

Let $I(\Omega)$ denote the radiative intensity traveling in a given direction, Ω. Different kinds of interactions are observed:

- Absorption: The medium absorbs a fraction of the incident radiation. The amount of absorbed radiation is $\kappa I(\Omega)$, where κ is the absorption coefficient.
- Emission: The medium emits radiation in all directions. The amount of emitted radiative intensity is equal to κI_b, where I_b is the *blackbody radiation intensity*.
- Scattering: Part of the radiation coming from a given direction is scattered in other directions. The scattering properties of the medium are described by the scattering phase function $\phi(\Omega', \Omega)$, which gives the probability that a ray coming from one direction Ω' is scattered into the direction Ω. The phase function $\phi(\Omega', \Omega)$ satisfies:

$$\frac{1}{4\pi}\int_{4\pi}\phi(\Omega', \Omega)d\Omega' = 1$$

Radiative intensity in a given direction is attenuated and augmented by scattering:

- It is attenuated because a part of the incident radiation in this direction is scattered into other directions. The amount of radiation attenuated by scattering is $\sigma_s I(\Omega)$.
- It is augmented because a part of the radiative intensity coming from other directions is scattered in all direction, including the direction we are looking at. The amount of radiation augmented by scattering is obtained by integrating scattering coming from all directions Ω':

$$\frac{\sigma_s}{4\pi}\int_{4\pi} I(\Omega')\phi(\Omega', \Omega)d\Omega'$$

Radiative Transfer Equation

The balance of the radiative intensity including all contributions (propagation, emission, absorption, and scattering) can now be formulated. The general radiative transfer equation can be written as (see Ref. 18):

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi}\int_{4\pi} I(\Omega')\phi(\Omega', \Omega)d\Omega' \tag{4-72}$$

where

- $I(\Omega)$ is the radiative intensity at a given position following the Ω direction (SI unit: $W/(m^2{\cdot}sr)$)
- $I_b(T)$ is the blackbody radiative intensity (SI unit: $W/(m^2{\cdot}sr)$), defined as

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi} \tag{4-73}$$

> The quantity $I_b(T)$ is available as a predefined function, `ht.fIb(T)`, in heat transfer interfaces.

- n_r is the refractive index (SI unit: 1)
- σ is the Stefan-Boltzmann constant (SI unit: $W/(m^2{\cdot}K^4)$)

- κ, β, σ_s are absorption, extinction, and scattering coefficients, respectively (SI unit: 1/m) and are related by:

$$\beta = \kappa + \sigma_s$$

- $\phi(\Omega', \Omega)$ is the scattering phase function (SI unit: 1)
- T is the temperature (SI unit: K)

The phase function, $\phi(\Omega', \Omega)$, gives the probability that a ray from the Ω' direction is scattered into the Ω direction. The phase function's definition is material dependent and its definition can be complicated. It is common to use approximate scattering phase functions that are defined using the cosine of the scattering angle, μ_0. The current implementation handles:

- *Isotropic* phase functions:

$$\phi(\Omega', \Omega) = \phi(\mu_0) = 1$$

- *Linear anisotropic* phase functions:

$$\phi(\mu_0) = 1 + a_1\mu_0$$

- *Polynomial anisotropic* up to the 12$^{\text{th}}$ order:

$$\phi(\mu_0) = 1 + \sum_{n=1}^{12} a_n P_n(\mu_0)$$

where P_n are the n-th order Legendre polynomials.

Legendre polynomials can be defined by the Rodriguez formula:

$$P_k(x) = \frac{1}{2^k k!}\frac{d^k}{dx^k}((x^2-1)^k)$$

A quantity of interest is the incident radiation, denoted G, and defined by

$$G = \int_{4\pi} I(\Omega)d\Omega$$

Boundary Condition for the Radiative Transfer Equation

For gray walls, corresponding to opaque surfaces reflecting diffusively and emitting, the radiative intensity $I(\Omega)$ entering participating media along the Ω direction is

$$I(\Omega) = \varepsilon I_b(T) + \frac{1-\varepsilon}{\pi} q_{r,\,out} \qquad \text{for all } \Omega \text{ such that } \mathbf{n}\cdot\Omega < 0$$

where

$$I_b(T) = \frac{n_r^{\,2}\sigma T^4}{\pi} \tag{4-74}$$

- Equation 4-73 is the blackbody radiation intensity and n_r is the refractive index
- ε is the surface emissivity, which is in the range [0, 1]
- $1-\varepsilon$ is the diffusive reflectivity
- $\mathbf{n}$ is the outward normal vector
- $q_{r,out}$ is the heat flux striking the wall:

$$q_{r,\,out} = \int_{\mathbf{n}\cdot\Omega > 0} I(\Omega)(\mathbf{n}\cdot\Omega)d\Omega$$

For black walls $\varepsilon = 1$. Thus $I(\Omega) = I_b(T)$.

Heat Transfer Equation in Participating Media

Heat flux in gray media is defined by

$$\mathbf{q}_r = \int_{4\pi} I(\Omega)\Omega d\Omega$$

Heat flux divergence can be defined as a function of G and T (see Ref. 18):

$$Q_r = \nabla\cdot\mathbf{q}_r = \kappa(G - 4\pi I_b(T))$$

In order to couple radiation in participating media, radiative heat flux is taken into account in addition to conductive heat flux. Recalling Equation 4-16, the heat transfer equation reads:

$$\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla T\right) + \nabla\cdot(\mathbf{q} + \mathbf{q}_r) = \alpha_p T\left(\frac{\partial p}{\partial t} + \mathbf{u}\cdot\nabla p\right) + \tau{:}\nabla\mathbf{u} + Q$$

and is implemented using following form:

$$\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla T\right) + \nabla\cdot\mathbf{q} = \kappa(G - 4n\sigma T^4) + \alpha_p T\left(\frac{\partial p}{\partial t} + \mathbf{u}\cdot\nabla p\right) + \tau{:}\nabla\mathbf{u} + Q$$

The *discrete ordinates method* is implemented for 3D and 2D geometries.

Radiative intensity is defined for any direction Ω, because the angular space is continuous. In order to handle the radiative intensity equation numerically, the angular space is discretized.

The discrete ordinates method (or S_N approximation) provides a discretization of angular space into $n = N(N+2)$ in 3D (or $n = N(N+2)/2$ in 2D) discrete directions. It consists in a set of directions and quadrature weights. Several sets are available in the literature. A set should satisfy first, second, and third moments (see Ref. 18); it is also recommended that the quadrature fulfills the half moment for vectors of Cartesian basis. Since it is not possible to fulfill exactly all these conditions, accuracy should be improved when N increases.

Following the conclusion of Ref. 19, the implementation uses the LSE symmetric quadrature for S2, S4, S6, and S8. The LSE symmetric quadrature fulfills the half, first, second, and third moments.

Thanks to angular space discretization, integrals over directions are replaced by numerical quadratures of discrete directions:

$$\int_{4\pi} I(\Omega)d\Omega \approx \sum_{j=1}^{n} w_j I_j$$

Depending on the value of N, a set of n dependent variables has to be defined and solved for $I_1,\ I_2,\ \ldots,\ I_n$.

Each dependent variable satisfies the equation

$$\mathbf{S}_i \cdot \nabla I_i \ = \ \kappa I_\mathrm{b}(T) - \beta I_i + \frac{\sigma_\mathrm{s}}{4\pi}\sum_{j=1}^{n} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i)$$

where $\mathbf{S}_i$ is the i-th discrete ordinate, with the following boundary condition

$$I_{i,\,\mathrm{bnd}} = \varepsilon I_{\mathrm{b}}(T) + \frac{1-\varepsilon}{\pi} q_{\mathrm{out}} \qquad \text{for all } \mathbf{S}_i \text{ such that } \mathbf{n}\cdot\mathbf{S}_i < 0$$

with

$$q_{\mathrm{r,\,out}} = \sum_{\mathbf{n}\cdot\Omega_j > 0} w_j I_j \mathbf{n}\cdot\Omega_j$$

Discrete Ordinates Method Implementation in 2D

For a given index i, define two indices, i^+ and i^-, so that

- Ω, $\mathbf{S}_{i+}$, and $\mathbf{S}_{i-}$ have the same components in the *xy*–plane
- and $\mathbf{S}_{i+}$ and $\mathbf{S}_{i-}$ have opposite components in the z direction.

Assuming that a model is invariant in the z direction, the radiative transfer equation in two directions, $\mathbf{S}_{i+}$ and $\mathbf{S}_{i-}$, for the discrete ordinates method (DOM) reads:

$$\mathbf{S}_{i^+}\cdot\nabla I_{i^+} = \kappa I_{\mathrm{b}}(T) - \beta I_{i^+} + \frac{\sigma_{\mathrm{s}}}{4\pi}\sum_{j=1}^{n} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i^+})$$

$$\mathbf{S}_{i^-}\cdot\nabla I_{i^-} = \kappa I_{\mathrm{b}}(T) - \beta I_{i^-} + \frac{\sigma_{\mathrm{s}}}{4\pi}\sum_{j=1}^{n} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i^-})$$

By summing these two equations and introducing $\tilde{I}_i$ which is equal to I_{i^-} and I_{i^+} (these are equal in 2D):

$$2\mathbf{S}_i\cdot\nabla\tilde{I}_i = 2\kappa I_{\mathrm{b}}(T) - 2\beta\tilde{I}_i + \frac{\sigma_{\mathrm{s}}}{4\pi}\sum_{j=1}^{n} w_j I_j(\phi(\mathbf{S}_j, \mathbf{S}_{i^+}) + \phi(\mathbf{S}_j, \mathbf{S}_{i^-}))$$

which can be rewritten as:

$$\mathbf{S}_i\cdot\nabla\tilde{I}_i = \kappa I_{\mathrm{b}}(T) - \beta\tilde{I}_i + \frac{\sigma_{\mathrm{s}}}{8\pi}\sum_{j=1}^{n} w_j I_j(\phi(\mathbf{S}_j, \mathbf{S}_{i^+}) + \phi(\mathbf{S}_j, \mathbf{S}_{i^-}))$$

In addition if $\phi(\mathbf{S}_i, \mathbf{S}_j)$ can be rewritten as a function of $\mathbf{S}_i\cdot\mathbf{S}_j$, as it is in COMSOL Multiphysics implementation, then

$$\phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^+}) = \phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^-}) \text{ and } \phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^+}) = \phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^-})$$

In addition

$$I_{j^-}\phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^+}) + I_{j^+}\phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^-}) = 2\tilde{I}_j\phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^-}) = 2\tilde{I}_j\phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^+})$$

so the above equation can be simplified:

$$\tilde{\mathbf{S}}_i \cdot \nabla\tilde{I}_i = \kappa I_\mathrm{b}(T) - \beta\tilde{I}_i + \frac{\sigma_\mathrm{s}}{4\pi}\sum_{j=1}^{n} w_j\tilde{I}_i\phi(\mathbf{S}_j, \mathbf{S}_i) \qquad \text{(4-75)}$$

with

$$\tilde{\mathbf{S}}_i = \begin{bmatrix} \mathbf{S}_{i,1} \\ \mathbf{S}_{i,2} \\ 0 \end{bmatrix}$$

since the third component of $\nabla\tilde{I}_i$ is zero in 2D.

Also notice that

$$\int_{4\pi} I(\Omega)d\Omega \approx \sum_{j=1}^{n} w_j I_j = \sum_{j=1}^{n/2} w_j I_{j^-} + w_{j^+} I_{j^+} = \sum_{j=1}^{n/2} \tilde{w}_i\tilde{I}_i \qquad \text{(4-76)}$$

with $\tilde{w}_i = 2w_i$.

Using results from Equation 4-75 and Equation 4-76 the DOM is formulated in 2D using only radiative intensities, $\tilde{I}_i$, on half of the 3D DOM directions, $\tilde{\mathbf{S}}_i$, except for the scattering term. In other expressions than the scattering term, the z component of the radiative intensities I_i and of the discrete directions Ω_i can be ignored (or set to zero) and the weight w_i, multiplied by 2.

Rosseland Approximation Theory

For The Heat Transfer with Radiation in Participating Media Interface, **Rosseland approximation** is available as a radiation discretization method. Then for Radiation in Participating Media (Heat Transfer Interface) feature node this theory is applicable.

Rosseland approximation relies on the hypotheses that the participating media is optically thick — that is, $\tau >> 1$ — where τ is the optical thickness defined by the integral of absorption coefficient, κ, along a typical optical path:

$$\tau = \int_0^s \kappa ds$$

From a computational point of view this approximation has a limited impact because it does not introduce any extra degree of freedom to the heat equation. Instead it adds nonlinear contribution to the thermal conductivity. This is why this method is popular for some applications where the optical thickness is large. Nevertheless, because it gives a simple approximation of heat transfer by radiation in a participating media, it should be carefully validated.

In this case, the radiative heat flux can be evaluated by (Ref. 18):

$$q_{\mathrm{r},\lambda} = -\frac{4\pi}{\beta_\lambda}\nabla i_{\mathrm{b},\lambda}$$

For a gray media it leads to

$$q_{\mathrm{r}} = -\frac{4\sigma}{3\beta_{\mathrm{R}}}\nabla(n^2T^4)$$

Assuming a constant refractive index, this can be rewritten as $q_{\mathrm{r}} = -k_{\mathrm{R}}\Delta T$ with

$$k_{\mathrm{R}} = \frac{16n^2\sigma T^3}{3\beta_{\mathrm{R}}}$$

and

$$q_{\mathrm{r}} = -\frac{16n^2\sigma T^3}{3\beta_R}\nabla T$$

Notice that the Rosseland approximation does not account at all for the scattering in the participating media.

P1 Approximation Theory

For The Heat Transfer with Radiation in Participating Media Interface and The Radiation in Participating Media Interface, **P1 approximation** is available as a radiation discretization method.

P1 approximation is the simplest approximation provided by the method of spherical harmonics method (PN-method). This approximation provides additional accuracy compared to a Rosseland approximation even if it remains a very simple method. The P1 method relies on the following hypotheses:

- The media is optically thick media: $\tau >> 1$, where τ is the optical thickness defined by the integral of absorption coefficient, κ, along a typical optical path:

$$\tau = \int_0^s \kappa ds$$

- The scattering is linear isotropic.

From a computational point of view this approximation has a limited impact because it introduces only one additional degree of freedom for G, which is a scalar quantity and adds a heat source or sink to the temperature equation to account for radiative heat transfer contributions. This method, however, fails to accurately represent cases where the radiative intensity propagation dominates over its diffusivity or where the scattering effects cannot be described by a linear isotropic phase function.

The P1 approximation accounts for the radiation transfer equation

$$\Omega \cdot \nabla I(\Omega) = \kappa I_{\mathrm{b}}(T) - \beta I(\Omega) + \frac{\sigma_{\mathrm{s}}}{4\pi}\int_{4\pi} I(\Omega')\phi(\Omega', \Omega) d\Omega'$$

by solving following equation for $G = \int_{4\pi} I(\Omega) d\Omega$ (Ref. 18):

$$-\nabla \cdot (D_{\mathrm{P1}} \nabla G) = Q_{\mathrm{r}} \tag{4-77}$$

where

- $D_{\mathrm{P1}}$is the P1 diffusion coefficient, defined as

$$D_{\mathrm{P1}} = \frac{1}{3\kappa + \sigma_s(3 - a_1)}$$

- a_1 is the linear Legendre coefficient of the scattering phase function
- Q_{r} is the radiative heat source:

$$Q_{\mathrm{r}} = \kappa(G - 4\pi I_{\mathrm{b}}) \tag{4-78}$$

When scattering is modeled as isotropic, a_1=0 and the P1 diffusion coefficient reduces to

$$D_{\mathrm{P1}} = \frac{1}{3\kappa + 3\sigma_s}$$

The following boundary condition applies (Ref. 18):

$$\mathbf{n} \cdot D_{\mathrm{P1}} \nabla G = -q_{\mathrm{r,\,net}}$$

where $\mathbf{q}_{\mathrm{r,\,net}}$ is the net radiative heat flux at the boundary.

RADIATION IN PARTICIPATING MEDIA

For the Radiation in Participating Media (Heat Transfer Interface) and Radiation in Participating Media (RPM Interface) feature nodes, the equation Equation 4-77 is implemented.

In addition Q_{r}, defined by Equation 4-78, is added as an heat source in the heat transfer equation:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot \mathbf{q} = Q_{\mathrm{r}} + \alpha_p T \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right) + \tau : \nabla \mathbf{u} + Q$$

OPAQUE SURFACE

The Opaque Surface boundary condition defines a boundary opaque to radiation and defines the incident intensity on a boundary:

$$\mathbf{n} \cdot D_{\mathrm{P1}} \nabla G = -\mathbf{q}_{\mathrm{r,\,net}}$$

The Opaque Surface feature accounts for the net radiative heat flux, $\mathbf{q}_{\mathrm{r,\,net}}$, in the heat balance.

Two cases are considered, depending on surface emissivity value:

- Gray wall: ε is a number between 0 and 1
- Black wall: ε=1

Gray Wall

The radiative heat flux at the boundary depends on the surface emissivity, ε:

$$q_{\mathrm{r,\,net}} = \frac{\varepsilon}{2(2-\varepsilon)}(4\pi I_{\mathrm{b,\,w}} - G)$$

with

$$I_{\mathrm{b,w}} = I_{\mathrm{b}} = \frac{n^2\sigma T^4}{\pi}$$

Black Wall

The radiative heat flux at the boundary expression simplifies to

$$\mathbf{n}\cdot D_{\mathrm{P1}}\nabla G = \frac{1}{2}(4\pi I_{\mathrm{b,w}} - G)$$

with

$$I_{\mathrm{b,w}} = I_{\mathrm{b}} = \frac{n^2\sigma T^4}{\pi}$$

INCIDENT INTENSITY

The Incident Intensity node defines a boundary that receives incident radiative intensity I_{ext} and that is transparent for outgoing intensity. On these boundaries, the relation between G, $\mathbf{q}_{\mathrm{r,net}}$ (net radiative heat flux) and I_{ext} (incident radiative intensity) is

$$G + 2q_{\mathrm{r,net}} = 4\int_{\Omega\cdot\mathbf{n}>0} I_{\mathrm{ext}}(\Omega)\Omega\cdot\mathbf{n}d\Omega$$

by defining

$$I_{\mathrm{ext}} = \int_{\Omega\cdot\mathbf{n}>0} I_{\mathrm{ext}}(\Omega)\Omega\cdot\mathbf{n}d\Omega$$

there is

$$q_{\mathrm{r,net}} = \frac{1}{2}(4I_{\mathrm{ext}} - G)$$

which defines the heat radiative heat flux and also contributes to G boundary condition:

$$\mathbf{n}\cdot D_{\mathrm{P1}}\nabla G = -\mathbf{q}_{\mathrm{r,net}}$$

Theory for Moisture Transport

The Moisture Transport Interface solves for the following equation derived from Ref. 13:

$$\xi\frac{\partial\phi}{\partial t}+\nabla\cdot(-\xi D_{\mathrm{w}}\nabla\phi-\delta_{\mathrm{p}}\nabla(\phi p_{\mathrm{sat}}(T))) = G \tag{4-79}$$

This equation models the moisture transfer as the sum of the capillary moisture flux:

$$-D_{\mathrm{w}}\nabla(w(\phi)) = -D_{\mathrm{w}}\frac{\partial w}{\partial\phi}\nabla\phi = -\xi D_{\mathrm{w}}\nabla\phi$$

and the vapor diffusion flux:

$$\delta_{\mathrm{p}}\nabla p_{\mathrm{v}}(T) = \delta_{\mathrm{p}}\nabla(\phi p_{\mathrm{sat}}(T))$$

with the following material properties, fields, and source:

- ξ (SI unit: $\mathrm{kg/m^3}$) is the moisture storage capacity.
- δ_{p} (SI unit: s) is the vapor permeability.
- ϕ (dimensionless) is the relative humidity.
- p_{sat} (SI unit: Pa) is the vapor saturation pressure.
- T (SI unit: K) is the temperature.
- D_{w} (SI unit: $\mathrm{m^2/s}$) is the moisture diffusivity.
- G (SI unit: $\mathrm{kg/m^3{\cdot}s}$) is the moisture source.

Theory for the Heat Transfer Multiphysics Couplings

In this section:

- Theory for the Non-Isothermal Flow and Conjugate Heat Transfer Interfaces
- Theory for the Thermoelectric Effect Interface
- Theory for the Local Thermal Non-Equilibrium Interface
- Theory for the Heat and Moisture Transport Interface
- Theory for the Electromagnetic Heating Interfaces
- Theory for the Thermal Stress Interface

Theory for the Non-Isothermal Flow and Conjugate Heat Transfer Interfaces

The following points of the theory of Non-Isothermal Flow and Conjugate Heat Transfer are discussed in this part:

- The Non-Isothermal Flow and Conjugate Heat Transfer Equations
- Turbulent Non-Isothermal Flow Theory
- Theory for the Non-Isothermal Screen Boundary Condition
- Theory for the Interior Fan Boundary Condition

See Theory for the Single-Phase Flow Interfaces and Theory for the Turbulent Flow Interfaces in the *CFD Module User's Guide* for a description of the theory related to laminar and turbulent single-phase flow interfaces.

THE NON-ISOTHERMAL FLOW AND CONJUGATE HEAT TRANSFER EQUATIONS

In industrial applications it is common that the density of a process fluid varies. These variations can have a number of different sources but the most common one is the presence of an inhomogeneous temperature field. This module includes the Non-Isothermal Flow predefined multiphysics coupling to simulate systems in which the density varies with temperature.

Other situations where the density might vary includes chemical reactions, for instance where reactants associate or dissociate.

The Non-Isothermal Flow and Conjugate Heat Transfer interfaces contain the fully compressible formulation of the continuity and momentum equations:

$$\begin{gathered}\frac{\partial\rho}{\partial t}+\nabla\cdot(\rho\mathbf{u}) = 0\\ \rho\frac{\partial\mathbf{u}}{\partial t}+\rho\mathbf{u}\cdot\nabla\mathbf{u} = -\nabla p+\nabla\cdot\tau+\mathbf{F}\end{gathered} \tag{4-80}$$

where

- ρ is the density (SI unit: kg/m^3)
- $\mathbf{u}$ is the velocity vector (SI unit: m/s)
- p is the pressure (SI unit: Pa)
- τ is the viscous stress tensor (SI unit: Pa), equal for a compressible fluid to:

$$\tau = \mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}})-\frac{2}{3}\mu(\nabla\cdot\mathbf{u})\mathbf{I}$$

- μ is the dynamic viscosity (SI unit: Pa·s)
- $\mathbf{F}$ is the body force vector (SI unit: N/m^3)

It also solves the heat equation, which for a fluid is given in Equation 4-16 by

$$\rho C_p\left(\frac{\partial T}{\partial t}+\mathbf{u}\cdot\nabla T\right)+\nabla\cdot(\mathbf{q}+\mathbf{q}_{\mathrm{r}}) = \alpha_p T\left(\frac{\partial p}{\partial t}+\mathbf{u}\cdot\nabla p\right)+\tau{:}\nabla\mathbf{u}+Q$$

where in addition to the quantities above

- C_p is the specific heat capacity at constant pressure (SI unit: J/(kg·K))
- T is the absolute temperature (SI unit: K)
- $\mathbf{q}$ is the heat flux by conduction (SI unit: W/m^2)
- $\mathbf{q}_{\mathrm{r}}$ is the heat flux by radiation (SI unit: W/m^2)
- α_p is the coefficient of thermal expansion (SI unit: 1/K):

$$\alpha_p = -\frac{1}{\rho}\frac{\partial\rho}{\partial T}$$

- Q contains heat sources other than viscous heating (SI unit: W/m^3)

The *work done by pressure changes* term

$$Q_p = \alpha_p T\left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p\right)$$

and the *viscous heating* term

$$Q_{\mathrm{vd}} = \tau : \nabla \mathbf{u}$$

are not included by default because they are usually negligible. These terms can, however, be added by selecting corresponding check boxes in the Non-Isothermal Flow feature.

The physics interface also supports heat transfer in solids (Equation 4-14):

$$\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u}_{\mathrm{trans}} \cdot \nabla T\right) + \nabla \cdot (\mathbf{q} + \mathbf{q}_{\mathrm{r}}) = Q_{\mathrm{ted}} + Q$$

where Q_{ted} is the thermoelastic damping heat source (SI unit: W/(m^3)). This term is not included by default but must be added by selecting the corresponding check box.

- The Heat Balance Equation
- Turbulent Non-Isothermal Flow Theory

TURBULENT NON-ISOTHERMAL FLOW THEORY

Turbulent energy transport is conceptually more complicated than energy transport in laminar flows because the turbulence is also a form of energy.

Equations for compressible turbulence are derived using the Favre average. The Favre average of a variable T is denoted $\tilde{T}$ and is defined by

$$\tilde{T} = \frac{\overline{\rho T}}{\bar{\rho}}$$

where the bar denotes the usual Reynolds average. The full field is then decomposed as

$$T = \tilde{T} + T''$$

With this notation the energy balance equation becomes

$$\frac{\partial}{\partial t}\left(\bar{\rho}\left(\tilde{E}+\frac{\tilde{u}_i\tilde{u}_i}{2}\right)+\frac{\overline{\rho u_i''u_i''}}{2}\right)+\frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\left(\tilde{H}+\frac{\tilde{u}_i\tilde{u}_i}{2}\right)+\tilde{u}_j\frac{\overline{\rho u_i''u_i''}}{2}\right) = \tag{4-81}$$

$$\frac{\partial}{\partial x_j}\left(-q_j-\overline{\rho u_j''H''}+\overline{\tau_{ij}u_i''}-\frac{\overline{\rho u_j''u_i''u_i''}}{2}\right)+\frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}}-\overline{\rho u_i''u_j''}))$$

where H is the enthalpy. The vector

$$q_j = -\lambda\frac{\partial T}{\partial x_j} \tag{4-82}$$

is the laminar conductive heat flux and

$$\tau_{ij} = \mu\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)-\frac{2}{3}\mu\frac{\partial u_k}{\partial x_k}\delta_{ij}$$

is the laminar, viscous stress tensor. Notice that the thermal conductivity is denoted λ.

The modeling assumptions are in large part analogous to those for incompressible turbulence modeling. The stress tensor

$$-\overline{\rho u_i''u''_j}$$

is modeled using the *Boussinesq approximation*:

$$-\overline{\rho u_i''u_j''} = \bar{\rho}\tau^{\mathrm{T}}{}_{ij} = \mu_{\mathrm{T}}\left(\frac{\partial\tilde{u}_i}{\partial x_j}+\frac{\partial\tilde{u}_j}{\partial x_i}\right)-\frac{2}{3}\mu\frac{\partial\tilde{u}_k}{\partial x_k}\delta_{ij}-\frac{2}{3}\bar{\rho}k\delta_{ij} \tag{4-83}$$

where k is the turbulent kinetic energy, which in turn is defined by

$$\bar{\rho}k = \frac{1}{2}\overline{\rho u_i''u_i''} \tag{4-84}$$

The correlation between u_j'' and H'' in Equation 4-81 is the turbulent transport of heat. It is modeled analogously to the laminar conductive heat flux

$$\overline{\rho u_j''H''} = q_j^{\mathrm{T}} = -\lambda_{\mathrm{T}}\frac{\partial\tilde{T}}{\partial x_j} = -\frac{\mu_{\mathrm{T}}C_p}{\mathrm{Pr}_{\mathrm{T}}}\frac{\partial\tilde{T}}{\partial x_j} \tag{4-85}$$

The molecular diffusion term,

$$\overline{\tau_{ij}u_i''}$$

and turbulent transport term,

$$\frac{\overline{\rho u_j'' u_i'' u_i''}}{2}$$

are modeled by a generalization of the molecular diffusion and turbulent transport terms found in the incompressible k equation

$$\overline{\tau_{ij} u_i''} - \frac{\overline{\rho u_j'' u_i'' u_i''}}{2} = \left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j} \tag{4-86}$$

Inserting Equation 4-82, Equation 4-83, Equation 4-84, Equation 4-85 and Equation 4-86 into Equation 4-81 gives

$$\begin{aligned}&\frac{\partial}{\partial t}\left(\bar{\rho}\left(\tilde{E} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k\right)\right) + \frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\left(\tilde{H} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k\right)\right) = \\ &\frac{\partial}{\partial x_j}\left(-q_j - q_j^T + \left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}} + \bar{\rho}\tau_{ij}^T))\end{aligned} \tag{4-87}$$

The Favre average can also be applied to the momentum equation, which, using Equation 4-83, can be written

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{u}_i) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{u}_i) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}(\overline{\tau_{ij}} + \bar{\rho}\tau_{ij}^T) \tag{4-88}$$

Taking the inner product between $\tilde{u}_i$ and Equation 4-88 results in an equation for the resolved kinetic energy, which can be subtracted from Equation 4-87 with the following result:

$$\begin{aligned}&\frac{\partial}{\partial t}(\bar{\rho}(\tilde{E} + k)) + \frac{\partial}{\partial x_j}(\rho\tilde{u}_j(\tilde{E} + k)) = \\ &-\bar{p}\frac{\partial \tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}\left(-q_j - q_j^T + \left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}} + \bar{\rho}\tau_{ij}^T))\end{aligned} \tag{4-89}$$

where the relation

$$\tilde{H} = \tilde{E} + \frac{\bar{p}}{\bar{\rho}}$$

has been used.

According to Wilcox (Ref. 22), it is usually a good approximation to neglect the contributions of k for flows with Mach numbers up to the supersonic range. This gives the following approximation of Equation 4-89:

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E})+\frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial \tilde{u}_j}{\partial x_j}+\frac{\partial}{\partial x_j}(-q_j-q_j^{\mathrm{T}})+\frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}}+\bar{\rho}\tau_{ij}^{\mathrm{T}})) \tag{4-90}$$

Larsson (Ref. 23) suggests to make the split

$$\overline{\tau_{ij}} = \tilde{\tau}_{ij}+\overline{\tau_{ij}''}$$

Since

$$\tilde{\tau}_{ij} \gg \overline{\tau_{ij}''}$$

for all applications of engineering interest, it follows that

$$\overline{\tau_{ij}} \approx \tilde{\tau}_{ij}$$

and consequently

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E})+\frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial \tilde{u}_j}{\partial x_j}+\frac{\partial}{\partial x_j}\left((\lambda+\lambda_{\mathrm{T}})\frac{\partial \tilde{T}}{\partial x_j}\right)+\frac{\partial}{\partial x_j}(\tilde{u}_i\tilde{\tau}_{ij}^{\mathrm{tot}}) \tag{4-91}$$

where

$$\tilde{\tau}_{ij}^{\mathrm{tot}} = (\mu+\mu_{\mathrm{T}})\left(\frac{\partial \tilde{u}_i}{\partial x_j}+\frac{\partial \tilde{u}_j}{\partial x_i}-\frac{2}{3}\frac{\partial \tilde{u}_k}{\partial x_k}\delta_{ij}\right)$$

Equation 4-91 is completely analogous to the laminar energy equation of Equation 4-13 and can be expanded using the same theory to get the temperature equation similar to Equation 4-16 (see for example Ref. 23):

$$\bar{\rho}C_p\frac{\partial \tilde{T}}{\partial t}+\bar{\rho}C_p\tilde{u}_j\frac{\partial \tilde{T}}{\partial x_j}+\frac{\partial}{\partial x_j}\left(-(\lambda+\lambda_{\mathrm{T}})\frac{\partial \tilde{T}}{\partial x_j}\right) = -\frac{1}{\bar{\rho}}\left(\frac{\partial \bar{\rho}}{\partial \tilde{T}}\right)_p\tilde{T}\left(\frac{\partial \bar{p}}{\partial t}+\tilde{u}_j\frac{\partial \bar{p}}{\partial x_j}\right)+\tilde{\tau}_{ij}\frac{\partial \tilde{u}_i}{\partial x_j}$$

which is the temperature equation solved in the turbulent Non-Isothermal Flow and Conjugate Heat Transfer interfaces.

Turbulent Conductivity

Kays-Crawford This is a relatively exact model for $\mathrm{Pr_T}$, while still quite simple. In Ref. 24, it is compared to other models for $\mathrm{Pr_T}$ and found to be a good approximation for most kinds of turbulent wall bounded flows except for turbulent flow of liquid metals. The model is given by

$$\mathrm{Pr_T} = \left(\frac{1}{2\mathrm{Pr_{T\infty}}} + \frac{0.3C_p\mu_\mathrm{T}}{\lambda\sqrt{\mathrm{Pr_{T\infty}}}} - \left(\frac{0.3C_p\mu_\mathrm{T}}{\lambda}\right)^2\left(1-\exp\left(-\frac{\lambda}{0.3C_p\mu_\mathrm{T}\sqrt{\mathrm{Pr_{T\infty}}}}\right)\right)\right)^{-1} \quad (4\text{-}92)$$

where the Prandtl number at infinity is $\mathrm{Pr_{T\infty}} = 0.85$ and λ is the conductivity.

Extended Kays-Crawford Weigand et al. (Ref. 25) suggested an extension of Equation 4-92 to liquid metals by introducing

$$\mathrm{Pr_{T\infty}} = 0.85 + \frac{100\lambda}{C_p\mu\mathrm{Re}_\infty^{0.888}}$$

where Re_∞, the Reynolds number at infinity must be provided either as a constant or as a function of the flow field. This is entered in the Model Inputs section of the Fluid feature.

Temperature Wall Functions

Analogous to the single-phase flow wall functions (see Wall Functions described for the Wall boundary condition), there is a theoretical gap between the solid wall and the computational domain for the fluid and temperature fields. This gap is often ignored when the computational geometry is drawn.

The heat flux between the fluid with temperature T_f and a wall with temperature T_w, is:

$$q_\mathrm{wf} = \frac{\rho C_p C_\mu^{1/4} k^{1/2}(T_\mathrm{w} - T_\mathrm{f})}{T^+}$$

where ρ is the fluid density, C_p is the fluid heat capacity, C_μ is a turbulence modeling constant, and k is the turbulent kinetic energy. T^+ is the dimensionless temperature and is given by (Ref. 26):

$$T^+ = \begin{cases} \mathrm{Pr}\,\delta_w^+ & \text{for } \delta_w^+ < \delta_{w1}^+ \\ 15\mathrm{Pr}^{2/3} - \dfrac{500}{\delta_w^{+2}} & \text{for } \delta_{w1}^+ \le \delta_w^+ < \delta_{w2}^+ \\ \dfrac{\mathrm{Pr}_T}{\kappa}\ln\delta_{w2}^+ + \beta & \text{for } \delta_{w2}^+ \le \delta_w^+ \end{cases}$$

where in turn

$$\delta_w^+ = \frac{\delta_w \rho \sqrt{C_\mu^{1/2} k}}{\mu} \qquad \delta_{w1}^+ = \frac{10}{\mathrm{Pr}^{1/3}}$$

$$\delta_{w2}^+ = 10\sqrt{10\frac{\kappa}{\mathrm{Pr}_T}} \qquad \mathrm{Pr} = \frac{C_p \mu}{\lambda}$$

$$\beta = 15\mathrm{Pr}^{2/3} - \frac{\mathrm{Pr}_T}{2\kappa}\left(1 + \ln\left(1000\frac{\kappa}{\mathrm{Pr}_T}\right)\right)$$

λ is the thermal conductivity, and κ is the von Karman constant equal to 0.41.

The computational results should be checked so that the distance between the computational fluid domain and the wall, δ_w, is everywhere small compared to any geometrical quantity of interest. The distance δ_w is available for evaluation on boundaries.

THEORY FOR THE NON-ISOTHERMAL SCREEN BOUNDARY CONDITION

When the Non-Isothermal Flow multiphysics coupling feature is active, the conditions that apply across a screen in isothermal flow are complemented by:

$$[H_0]_-^+ = 0 \tag{4-93}$$

where H_0 is the total enthalpy.

- See Screen for the feature node details.
- Also see Screen Boundary Condition described for the single-phase flow interfaces.

THEORY FOR THE INTERIOR FAN BOUNDARY CONDITION

When the Non-Isothermal Flow multiphysics coupling feature is active, the conditions that apply across an interior fan are complemented by:

- If direction is Along normal vector, the outlet temperature is `T_avg` where `T_avg` is the weighted averaged temperature defined as:

$$T_{\mathrm{avg}} = \frac{\int_{\Gamma} (\mathrm{down}(\mathbf{u}\cdot\mathbf{n}\rho C_p T))dS}{\max\left(\varepsilon, \int_{\Gamma} (\mathrm{down}(\mathbf{u}\cdot\mathbf{n}\rho C_p T))dS\right)}$$

- If the direction is opposite to normal vector, the outlet temperature is `T_avg` where `T_avg` is:

$$T_{\mathrm{avg}} = \frac{\int_{\Gamma} (\mathrm{up}(\mathbf{u}\cdot\mathbf{n}\rho C_p T))dS}{\max\left(\varepsilon, \int_{\Gamma} (\mathrm{up}(\mathbf{u}\cdot\mathbf{n}\rho C_p T))dS\right)}$$

Theory for the Thermoelectric Effect Interface

The Thermoelectric Effect Interface implements thermoelectric effect, which is the direct conversion of temperature differences to electric voltage or vice versa. Devices such as thermoelectric coolers for electronic cooling or portable refrigerators rely on this effect. While Joule heating (resistive heating) is an irreversible phenomenon, the thermoelectric effect is, in principle, reversible.

Historically, the thermoelectric effect is known by three different names, reflecting its discovery in experiments by Seebeck, Peltier, and Thomson. The *Seebeck effect* is the conversion of temperature differences into electricity, the *Peltier effect* is the conversion of electricity to temperature differences, and the *Thomson effect* is heat produced by the product of current density and temperature gradients. These effects are thermodynamically related by the Thomson relations:

$$P = ST$$

$$\mu_{\mathrm{Th}} = T\frac{dS}{dT}$$

where P is the Peltier coefficient (SI unit: V), S is the Seebeck coefficient (SI unit: V/K), T is the temperature (SI unit: K), and μ_{Th} is the Thomson coefficient (SI unit: V/K). These relations show that all coefficients can be considered different descriptions of one and the same quantity. The COMSOL formulation primarily uses the Seebeck coefficient. The Peltier coefficient is also used as an intermediate variable, but the Thomson coefficient is not used.

When simulating the thermoelectric effect, the following fluxes are the quantities of interest:

- Conductive heat flux $\mathbf{q}$, defined by

$$\mathbf{q} = -k\nabla T + P\mathbf{J} \tag{4-94}$$

- Electric current density $\mathbf{J}$, defined by

$$\mathbf{J} = -\sigma(\nabla V + S\nabla T) \tag{4-95}$$

Thermoelectric efficiency is measured by the figure of merit Z (SI unit: 1/K), defined as:

$$Z = \frac{S^2\sigma}{k}$$

where σ is the electrical conductivity and k the thermal conductivity.

Some other quantities of relevance are the electric field $\mathbf{E}$ and the Joule heat source Q:

$$\mathbf{E} = -\nabla V$$

$$Q = \mathbf{J}\cdot\mathbf{E}$$

From these definitions, conservation of heat energy and electrical current in an immobile solid reads:

$$\rho C_p\frac{\partial T}{\partial t} + \nabla\cdot\mathbf{q} = Q$$

$$\nabla\cdot\mathbf{J} = Q_j$$

where ρ is the density, C_p the heat capacity, and Q_j is the current source.

HOW THE SEEBECK, PELTIER, AND THOMSON EFFECTS ARE INCLUDED IN THE GENERAL FORMULATION

The general formulation of thermoelectric effect redefines the heat flux and the electric current according to Equation 4-94 and Equation 4-95, respectively. This formulation does not necessarily correspond to the formulation used when only a particular aspect of thermoelectric effect is considered: Seebeck, Peltier, or Thomson. This paragraph describes how these separated effects can be recognized in the general formulation.

Seebeck Effect

The Seebeck effect is described as the conversion of temperature gradient into electric current. The contribution of the Seebeck effect is defined as a current contribution

$$\mathbf{J}_{\mathrm{Se}} = -\sigma S \nabla T$$

This formulation corresponds directly to Equation 4-95 used in the general formulation.

Peltier Effect

The Peltier effect is described as the conversion of t electric current in heat source or sink. It is defined as an heat source contribution

$$Q_{\mathrm{Pe}} = -P \nabla \cdot \mathbf{J}$$

This contribution is obtained by developing the divergence of $\mathbf{q}$ term in the heat equation when $\mathbf{q}$ is defined following Equation 4-94.

Thomson Effect

The Thomson effect defines the heat source induced by a current in presence of a temperature gradient in thermoelectric material. The heat source is defined by

$$Q_{\mathrm{Th}} = -\mu_{\mathrm{Th}} \mathbf{J} \cdot \nabla T$$

This contribution is obtained again by developing the divergence of the q term in the heat equation when $\mathbf{q}$ is defined following Equation 4-94. This time consider the term $-T\mathbf{J} \cdot \nabla S$. Assuming that S is function of T, then:

$$-T\mathbf{J} \cdot \nabla S = -T\frac{dS}{dT}\mathbf{J} \cdot \nabla T = -\mu_{\mathrm{Th}} \mathbf{J} \cdot \nabla T$$

Theory for the Local Thermal Non-Equilibrium Interface

The detailed theory leading to the equations of local thermal non-equilibrium heat transfer in porous media is presented above in Theory for Heat Transfer in Porous Media. This part only recalls the main results and describes how Local Thermal Non-Equilibrium multiphysics coupling feature implements them.

The local thermal non-equilibrium hypothesis describes heat transfer in a porous medium using two temperature fields to solve: T_f for the fluid phase and T_s for the porous matrix. These should satisfy the following couple of partial differential equations:

$$\theta_p \rho_s C_{p,s} \frac{\partial T_s}{\partial t} + \nabla \cdot \mathbf{q}_s = q_{sf}(T_f - T_s) + \theta_p Q_s \tag{4-96}$$

$$(1-\theta_p)\rho_f C_{p,f} \frac{\partial T_f}{\partial t} + \rho_f C_{p,f} \mathbf{u}_p \cdot \nabla T_f + \nabla \cdot \mathbf{q}_f = q_{sf}(T_s - T_f) + (1-\theta_p) Q_f \tag{4-97}$$

Recall the Fourier's law of conduction adapted to the local thermal non-equilibrium hypothesis:

$$\mathbf{q}_s = -\theta_p k_s \nabla T_s$$

$$\mathbf{q}_f = -(1-\theta_p) k_f \nabla T_f$$

and the quantities used in this problem:

- θ_p is the solid volume fraction (SI unit: 1)
- ρ_s and ρ_f are the solid and fluid densities (SI unit: kg/m^3)
- $C_{p,s}$ and $C_{p,f}$ are the solid and fluid heat capacities at constant pressure (SI unit: J/(kg·K))
- $\mathbf{q}_s$ and $\mathbf{q}_f$ are the solid and fluid conductive heat fluxes (SI unit: W/m^2)
- k_s and k_f are the solid and fluid thermal conductivities (SI unit: W/(m·K))
- q_{sf} is the interstitial convective heat transfer coefficient (SI unit: $W/(m^3 \cdot K)$)
- Q_s and Q_f are the solid and fluid heat sources (SI unit: W/m^3)
- $\mathbf{u}_p$ is the porous velocity vector (SI unit: m/s)

PREDEFINED MULTIPHYSICS INTERFACE

The Local Thermal Non-Equilibrium Interface is a predefined coupling between The Heat Transfer in Solids Interface and The Heat Transfer in Fluids Interface. These two interfaces solve for Equation 4-96 and Equation 4-97, respectively, but without the heat exchange term $\pm q_{sf}(T_f - T_s)$.

The Local Thermal Non-Equilibrium multiphysics coupling feature combines two actions in order to couple the two aforementioned physics interfaces. It first multiplies each energy equation by its volume fraction: θ_p and $(1-\theta_p)$ for solid and fluid phases, respectively. Then it adds the heat exchange term $\pm q_{sf}(T_f - T_s)$ in both equations.

VOLUMETRIC AND SURFACE THERMAL CONDITIONS

As shown in Equation 4-96 and Equation 4-97, the volumetric heat sources $\theta_p Q_s$ and $(1-\theta_p)Q_f$ are applied to the energy equations. The Heat Source features of each physics interface though specifies Q_s and Q_f. Special care is therefore needed when defining a heat source for the whole porous medium. You would have to ensure that the heat source densities, Q_s and Q_f, are both equal to the heat rate density that was intended to the porous medium.

Theory for the Heat and Moisture Transport Interface

The Heat and Moisture multiphysics coupling implements the following equations for heat and moisture transport, derived from Ref. 13:

$$(\rho C_p)_{\mathrm{eff}}\frac{\partial T}{\partial t} + \nabla\cdot(-k_{\mathrm{eff}}\nabla T - L_v\delta_p\nabla(\phi p_{\mathrm{sat}})) = Q \tag{4-98}$$

$$\xi\frac{\partial\phi}{\partial t} + \nabla\cdot(-\xi D_w\nabla\phi - \delta_p\nabla(\phi p_{\mathrm{sat}})) = G \tag{4-99}$$

where:

- $(\rho C_p)_{\mathrm{eff}}$ (SI unit: J/(m^3·K)) is the effective volumetric heat capacity at constant pressure.
- T (SI unit: K) is the temperature.
- k_{eff} (SI unit: W/(m·K)) is the effective thermal conductivity.
- L_v (SI unit: J/kg) is the latent heat of evaporation.
- δ_p (SI unit: s) is the vapor permeability.
- ϕ (dimensionless) is the relative humidity.
- p_{sat} (SI unit: Pa) is the vapor saturation pressure.
- Q (SI unit: W/m^3) is the heat source.
- ξ (SI unit: kg/m^3) is the moisture storage capacity.

- D_w (SI unit: m^2/s) is the moisture diffusivity.
- G (SI unit: W/m^3) is the moisture source.

Theory for the Electromagnetic Heating Interfaces

COMSOL Multiphysics provides four multiphysics interfaces for handling electromagnetic heating in the Heat Transfer interfaces:

- The Joule Heating Interface
- The Laser Heating Interface (requires the Wave Optics Module)
- The Induction Heating Interface (requires the AC/DC Module for 3D models)
- The Microwave Heating Interface (requires the RF Module)

They all have in common these three multiphysics coupling features:

- Electromagnetic Heat Source
- Boundary Electromagnetic Heat Source
- Temperature Coupling

The first two features add weak contributions due to resistive losses in the domains and boundaries, respectively. The underlying theory can be found in the *AC/DC Module User's Guide*, *RF Module User's Guide*, and *Wave Optics Module User's Guide*.

The reverse coupling is carried by the Temperature Coupling feature that shares the temperature variable with the electromagnetics interfaces.

Theory for the Thermal Stress Interface

In the Structural Mechanics interfaces, two multiphysics interfaces handle thermal stress:

- The Thermal Stress Interface
- The Joule Heating and Thermal Expansion Interface

Both require the Structural Mechanics Module. They have in common the use of the Thermal Expansion multiphysics coupling that models temperature dependence of the strain tensor and thermoelastic damping. For more details about the underlying theory, see the *Structural Mechanics Module User's Guide*.

Theory for Thermal Contact

The Thermal Contact feature has correlations to evaluate the joint conductance at two contacting surfaces.

The heat fluxes at the upside and downside boundaries depend on the temperature difference according to the relations:

$$-\mathbf{n}_{\mathrm{d}} \cdot (-k_{\mathrm{d}} \nabla T_{\mathrm{d}}) = -h(T_{\mathrm{u}} - T_{\mathrm{d}}) + rQ_{\mathrm{b}}$$

$$-\mathbf{n}_{\mathrm{u}} \cdot (-k_{\mathrm{u}} \nabla T_{\mathrm{u}}) = -h(T_{\mathrm{d}} - T_{\mathrm{u}}) + (1-r)Q_{\mathrm{b}}$$

At a microscopic level, contact is made at a finite number of spots as in Figure 4-13.

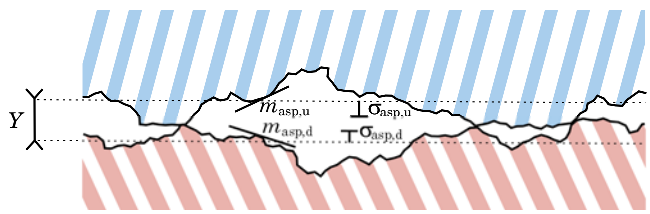


Figure 4-13: Contacting surfaces at the microscopic level.

The joint conductance, h, has three contributions: the constriction conductance, h_{c}, from the contact spots, the gap conductance, h_{g}, due to the fluid at the interstitial space, and the radiative conductance, h_{r}:

$$h = h_{\mathrm{c}} + h_{\mathrm{g}} + h_{\mathrm{r}}$$

Surface Asperities

The microscopic surface asperities are characterized by the average height $\sigma_{\mathrm{u,\,asp}}$ and $\sigma_{\mathrm{d,\,asp}}$ and the average slope $m_{\mathrm{u,\,asp}}$ and $m_{\mathrm{d,\,asp}}$. The RMS values σ_{asp} and m_{asp} are (4.16 in Ref. 20):

$$\sigma_{\mathrm{asp}} = \sqrt{\sigma_{\mathrm{u,\,asp}}^2 + \sigma_{\mathrm{d,\,asp}}^2} \qquad m_{\mathrm{asp}} = \sqrt{m_{\mathrm{u,\,asp}}^2 + m_{\mathrm{d,\,asp}}^2}$$

COOPER-MIKIC-YOVANOVICH (CMY) CORRELATION

The Cooper-Mikic-Yovanovich (CMY) correlation is valid for isotropic rough surfaces and has been formulated using a model assuming plastic deformation of the surface asperities. However, this model does not compute nor store the plastic deformations of the asperities. It means that, despite that a plastic deformation of the asperities is assumed, this contact model has no memory. For example, if a load is applied twice the thermal contact is identical in both cases. The Cooper-Mikic-Yovanovich (CMY) correlation relates h_c to the asperities and pressure load at the contact interface:

$$h_c = 1.25 k_{\mathrm{contact}} \frac{m_{\mathrm{asp}}}{\sigma_{\mathrm{asp}}} \left(\frac{p}{H_c}\right)^{0.95}$$

Here, H_c is the microhardness of the softer material, p is the contact pressure, and k_{contact} is the harmonic mean of the contacting surface conductivities:

$$k_{\mathrm{contact}} = \frac{2 k_u k_d}{k_u + k_d}$$

When k_u (resp. k_d) is not isotropic, it is replaced by its normal conductivity $\mathbf{n}^T k_u \mathbf{n}$ (resp. $\mathbf{n}^T k_d \mathbf{n}$).

The relative pressure p/H_c can be evaluated by specifying H_c directly or using the following relation (4.16.1 in Ref. 20) for the relative pressure using the Vickers correlation coefficient c_1 and size index c_2:

$$\frac{p}{H_c} = \left(\frac{p}{c_1 \left(1.62 \frac{\sigma_{\mathrm{asp}}}{\sigma_0} m_{\mathrm{asp}}\right)^{c_2}}\right)^{\frac{1}{(1 + 0.071 c_2)}}$$

where σ_0 is equal to 1 µm. For materials with a Brinell hardness between 1.30 and 7.60 GPa, c_1 and c_2 are given by the correlation below (4.16.1 in Ref. 20):

$$\frac{c_1}{H_0} = 4.0 - 5.77 \frac{H_B}{H_0} + 4.0 \left(\frac{H_B}{H_0}\right)^2 - 0.61 \left(\frac{H_B}{H_0}\right)^3$$

$$c_2 = -0.37 + 0.442\frac{H_B}{c_1}$$

The Brinell hardness is denoted by H_B, and H_0 is equal to 3.178 GPa.

MIKIC ELASTIC CORRELATION

The Mikic correlation is valid for isotropic rough surfaces and assumes elastic deformations of surface asperities. It gives h_c by the following relation:

$$h_c = 1.54k_{\text{contact}}\frac{m_{\text{asp}}}{\sigma_{\text{asp}}}\left(\frac{\sqrt{2}p}{mE_{\text{contact}}}\right)^{0.94}$$

Here, E_{contact} is an effective Young's modulus for the contact interface, satisfying (4.16.3 in Ref. 20):

$$\frac{1}{E_{\text{contact}}} = \frac{1-\nu_u^2}{E_u} + \frac{1-\nu_d^2}{E_d}$$

where E_u and E_d are the Young's moduli of the two contacting surfaces and ν_u and ν_d are the Poisson's ratios.

Gap Conductance

The gap conductance due to interstitial fluid cannot be neglected for high fluid thermal conductivity or high contact pressure. The parallel-plate gap gas correlation assumes that the interstitial fluid is a gas and defines h_g by:

$$h_g = \frac{k_{\text{gap}}}{Y + M_g}$$

Here k_{gap} is the gas conductivity, Y denotes the mean separation thickness (see Figure 4-13), and M_g is the gas parameter equal to:

$$M_g = \alpha\beta\Lambda \qquad \Lambda = \frac{k_B T_g}{\sqrt{2}\pi D^2 p_g}$$

In these relations, α is the contact thermal accommodation parameter, β is a gas property parameter (equal to 1.7 for air), Λ is the gas mean free path, k_B is the Boltzmann constant, D is the average gas particle diameter, p_g is the gas pressure (often the atmospheric pressure), and T_g is the gap temperature equal to:

$$T_g = \frac{T_u + T_d}{2}$$

The mean separation thickness, Y, is a function of the contact pressure, p. For low values of p near 0 Pa, Y goes to infinity since no contact occur. For high values of p — greater than $H_c/2$ in the Cooper-Mikic-Yovanovich model and greater than $H_c/4$ in the Mikic elastic model — Y reduces to 0 meaning that the contact is considered as perfect.

Radiative Conductance

At high temperatures, above 600 °C, radiative conductance needs to be considered. The gray-diffuse parallel plate model provides the following formula for h_r:

$$h_r = \frac{\varepsilon_u \varepsilon_d}{\varepsilon_u + \varepsilon_d - \varepsilon_u \varepsilon_d} \sigma(T_u^3 + T_u^2 T_d + T_u T_d^2 + T_d^3)$$

which implies that:

$$h_r(T_u - T_d) = \frac{\varepsilon_u \varepsilon_d}{\varepsilon_u + \varepsilon_d - \varepsilon_u \varepsilon_d} \sigma(T_u^4 - T_d^4)$$

$$h_r(T_d - T_u) = \frac{\varepsilon_u \varepsilon_d}{\varepsilon_u + \varepsilon_d - \varepsilon_u \varepsilon_d} \sigma(T_d^4 - T_u^4)$$

Thermal Friction

The friction heat source, Q_b, is partitioned into rQ_b and $(1 - r)Q_b$ at the contact interface. If the two bodies are identical, r and $(1 - r)$ would be 0.5 so that half of the friction heat goes to each surface. However, in the general case where the two bodies are made of different materials, the partition rate might not be 0.5. The Charron's relation (Ref. 21) defines r as:

$$r = \frac{1}{1 + \xi_d} \qquad \xi_d = \sqrt{\frac{\rho_u C_{p,\,u} k_u}{\rho_d C_{p,\,d} k_d}}$$

and symmetrically, $(1 - r)$ is:

$$(1 - r) = \frac{1}{1 + \xi_u} \qquad \xi_u = \sqrt{\frac{\rho_d C_{p,\,d} k_d}{\rho_u C_{p,\,u} k_u}}$$

For anisotropic conductivities, $\mathbf{n}^{\mathrm{T}}k_{\mathrm{d}}\mathbf{n}$ and $\mathbf{n}^{\mathrm{T}}k_{\mathrm{u}}\mathbf{n}$ replaces k_{d} and k_{u}, respectively.

Thermal Contact

Contact Switch: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/contact_switch**

Moist Air Fluid Type

For The Heat Transfer in Fluids Interface, you can select moist air as the fluid type. This is provided to calculate the relative humidity and to deduce if there is condensation. The following theory assumes that moist air is an ideal gas.

Humidity

This part defines the different definitions of humidity in the moist air theory.

MOISTURE CONTENT

The *moisture content* (also called *mixing ratio* or *humidity ratio*) is defined as the ratio of water vapor mass, m_{v}, to dry air mass, m_{a}:

$$x_{\mathrm{vap}} = \frac{m_{\mathrm{v}}}{m_{\mathrm{a}}} = \frac{p_{\mathrm{v}} M_{\mathrm{v}}}{p_{\mathrm{a}} M_{\mathrm{a}}} \tag{4-100}$$

where p_{v} is the water vapor partial pressure, p_{a} is the dry air partial pressure, and M_{a} and M_{v} are the molar mass of dry air and water vapor, respectively. Without condensation, the moisture content is not affected by temperature and pressure. The moisture content represents a ratio of mass, and it is thus a dimensionless number.

RELATIVE HUMIDITY

The *relative humidity* of an air mixture is expressed as follows:

$$\phi = \frac{p_{\mathrm{v}}}{p_{\mathrm{sat}}} \tag{4-101}$$

where p_{v} is the water vapor partial pressure and p_{sat} is the saturation pressure of water vapor.

According to Dalton's law, the total pressure of a mixture of gases is the sum of all the partial pressures of each individual gas; that is, $p = p_{\mathrm{v}} + p_{\mathrm{a}}$ where p_{a} is the dry air partial pressure.

The relative humidity formulation is often used to quantify humidity. However, for the same quantity of moisture content, the relative humidity changes with temperature and pressure, so in order to compare different values of ϕ it has to be at the same temperature and pressure conditions.

This quantity is useful to study the condensation as it defines the boundary between the liquid phase and the vapor phase. In fact, when the relative humidity ϕ reaches unity, it means that the vapor is saturated and that water vapor condenses.

The **Reference relative humidity** (dimensionless) is a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to a water vapor-saturated air. This **Reference relative humidity** associated to the **Reference temperature** and the **Reference pressure** are used to calculate the moisture content. Then the thermodynamical properties of moist air can be deduced through the mixture formula described below.

> The **Reference relative humidity** cannot be greater than 1, above which value the water vapor is condensing. If the value is greater than 1, the **Reference relative humidity** value is forced to be 1. The condensation area cannot be simulated.

SPECIFIC HUMIDITY

The *specific humidity* is defined as the ratio of water vapor, m_v, to the total mass, $m_{tot} = m_v + m_a$:

$$\omega = \frac{m_v}{m_{tot}} \tag{4-102}$$

> When the water vapor only accounts for a few percent in the total mass, the moisture content and the specific humidity are very close: $x_{vap} \approx \omega$ (only for low values). For larger values of ω, the two quantities are more precisely related by:
>
> $$x_{vap} = \frac{\omega}{1-\omega}$$

CONCENTRATION

The concentration is defined by:

$$c_v = \frac{n_v}{V} \tag{4-103}$$

where n_v is the amount of water vapor (SI unit: mol) and V is the total volume (SI unit: m^3). According to the ideal gas hypothesis, the saturation concentration is defined as follows:

$$c_{sat} = \frac{p_{sat}(T)}{RT} \tag{4-104}$$

Saturation State

The saturation state is reached when the relative humidity reaches one. It means that the partial pressure of the water vapor is equal to the saturation pressure (which also depends on the temperature).

From Ref. 31, the saturation pressure can be defined using the following expression:

$$p_{sat}(T) = 610.7[\mathrm{Pa}] \cdot 10^{7.5\frac{T - 273.15[\mathrm{K}]}{T - 35.85[\mathrm{K}]}} \tag{4-105}$$

Temperature and saturation pressure are deduced from this formulation.

Moist Air Properties

The thermodynamical properties of moist air can be found with some mixture laws. These are defined in this paragraph.

PRELIMINARY DEFINITIONS

Molar Fraction

The molar fraction of dry air, $X_{a,}$ and the molar fraction of water vapor, $X_{v,}$ are defined such as:

$$X_a = \frac{n_a}{n_{tot}} = \frac{p_a}{p} = \frac{p - \phi p_{sat}}{p} \tag{4-106}$$

$$X_v = \frac{n_v}{n_{tot}} = \frac{p_v}{p} = \frac{\phi p_{sat}}{p} \tag{4-107}$$

where:

- n_a is amount of dry air
- n_v is amount of water vapor
- n_{tot} is the total amount of moist air in mol
- p_a is the partial pressure of dry air
- p_v is the partial pressure of water vapor

- p is the pressure
- ϕ is the relative humidity, and
- p_{sat} is the saturation pressure.

From Equation 4-106 and Equation 4-107, the following relation holds:

$$X_a + X_v = 1$$

Relative Humidity and Moisture Content

Moisture content and relative humidity can be related with the following expression:

$$\phi = \frac{x_{vap}p}{p_{sat}\left(\frac{M_v}{M_a} + x_{vap}\right)} \tag{4-108}$$

MIXTURE PROPERTIES

The thermodynamical properties are built through a mixture formula. The expressions depend on dry air properties and pure steam properties and are balanced by the mass fraction.

Density

According to the ideal gas law, the mixture density ρ_m expression is defined as follows:

$$\rho_m = \frac{p}{RT}(M_a X_a + M_v X_v) \tag{4-109}$$

where M_a and M_v are the molar mass of dry air and water vapor, respectively, and X_a and X_v are the molar fraction of dry air and water vapor, respectively.

The ideal gas assumption sets the compressibility factor and the enhancement factor to unity. In fact, the accuracy lost by this assumption is small as the pure steam represents a small fraction.

Heat Capacity at Constant Pressure

According to Ref. 32, the heat capacity at constant pressure of a mixture is:

$$C_{p,m} = \frac{M_a}{M_m}X_a C_{p,a} + \frac{M_v}{M_m}X_v C_{p,v} \tag{4-110}$$

where M_m represents the mixture molar fraction and is defined by

$$M_{\mathrm{m}} = X_{\mathrm{a}}M_{\mathrm{a}} + X_{\mathrm{v}}M_{\mathrm{v}}$$

and where $C_{p,\,\mathrm{a}}$ and $C_{p,\,\mathrm{v}}$ are the heat capacity at constant pressure of dry air and steam, respectively.

Dynamic Viscosity

According to Ref. 32 and Ref. 33, the dynamic viscosity is defined as:

$$\mu_{\mathrm{m}} = \sum_{i\,=\,\mathrm{a,\,v}} \frac{X_i\mu_i}{\sum_{j\,=\,a,\,\mathrm{v}} X_j\varphi_{ij}} \tag{4-111}$$

where φ_{ij} is given by

$$\varphi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j}\right)^{\frac{1}{2}}\left(\frac{M_j}{M_i}\right)^{\frac{1}{4}}\right]^2}{\left[8\left(1 + \frac{M_i}{M_j}\right)\right]^{\frac{1}{2}}}$$

Here, μ_{a} and μ_{v} are the dynamic viscosity of dry air and steam, respectively.

Thermal Conductivity

According to Ref. 33 and Ref. 32, the thermal conductivity of the mixture is defined similarly:

$$k_{\mathrm{m}} = \sum_{i\,=\,\mathrm{a,\,v}} \frac{X_ik_i}{\sum_{j\,=\,\mathrm{a,\,v}} X_j\varphi_{ij}} \tag{4-112}$$

where k_{a} and k_{v} are the thermal conductivity of dry air and steam, respectively.

PURE COMPONENT PROPERTIES

The dry air and steam properties used to define the mixture properties are temperature-dependent high-order polynomials. The polynomials have been computed according to Ref. 16 for dry air properties and Ref. 34 for pure steam properties. The steam properties are based on the Industrial Formulation IAPWS-IF97.

The valid temperature range is $200\ \mathrm{K} < T < 1200\ \mathrm{K}$ for dry air properties and $273.15\ \mathrm{K} < T < 873.15\ \mathrm{K}$ for steam properties.

RESULTS AND ANALYSIS VARIABLES

These variables are provided to display the related quantities:

- Moisture content `xvap`.
- Vapor mass fraction `omega_moist`.
- Concentration of water vapor `c`.
- Relative humidity `phi`. This variable corresponds to the calculated ϕ with the system temperature and pressure.
- Condensation indicator `condInd`; this indicator is set to 1 if condensation has been detected ($\phi = 1$) and 0 if not.

FUNCTIONS

The following functions are defined and can be used as feature parameters as well as in postprocessing. Here, *feature* stands for `fluid` or `porous`, depending on whether the function is defined in the **Fluid** or in the **Porous Medium** feature:

- `ht.`*feature*`.fc(RH,T, pA)`, where `RH` is the relative humidity $0 \le \phi \le 1$, `T` is the temperature (SI unit: K), and `pA` is the pressure (SI unit: Pa). It returns the corresponding water vapor concentration (SI unit: mol/m^3) by deriving the following relation from Equation 4-101, Equation 4-104, and Equation 4-108:

$$c_{\mathrm{v}} = \frac{x_{\mathrm{vap}}p}{\left(x_{\mathrm{vap}} + \frac{M_{\mathrm{v}}}{M_{\mathrm{a}}}\right)RT}$$

	The concentration computation assumes that the ideal gas assumption is valid.

- `ht.`*feature*`.fxvap(RH, T, pA)`, where `RH` is the relative humidity $0 \le \phi \le 1$, `T` is the temperature (SI unit: K) and `pA` is the pressure (SI unit Pa). It returns the moisture content (SI unit: 1) by using the following relation:

$$x_{\mathrm{vap}} = \frac{\phi p_{\mathrm{sat}}}{p - \phi p_{\mathrm{sat}}} \cdot \frac{M_{\mathrm{v}}}{M_{\mathrm{a}}}$$

- `ht.`*feature*`.fpsat(T)`, where `T` is the temperature (SI unit: K). It returns the saturation pressure (SI unit: Pa) by using Equation 4-105.
- `ht.`*feature*`.Lv(T)`, where `T` is the temperature (SI unit: K). It returns the latent heat of evaporation (SI unit: J/kg) as a linear interpolation of the data from Ref. 34,

which provides steam properties based on the Industrial Formulation IAPWS-IF97. The temperature-dependency is as shown on Figure 4-14.

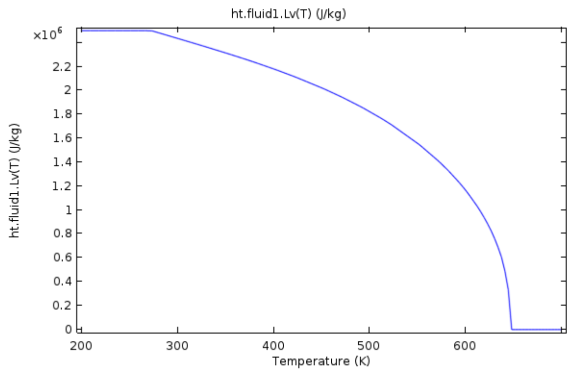


Figure 4-14: Latent heat of evaporation in function of temperature

Out-of-Plane Heat Transfer

When the object to model in COMSOL Multiphysics is thin or slender enough along one of its geometry dimensions, there is usually only a small variation in temperature along the object's thickness or cross section. For such objects, it is computationally more efficient to reduce the model geometry to 2D or even 1D and to use the out-of-plane heat transfer mechanism. Figure 4-15 shows examples of possible situations in which this type of geometry reduction can be applied.

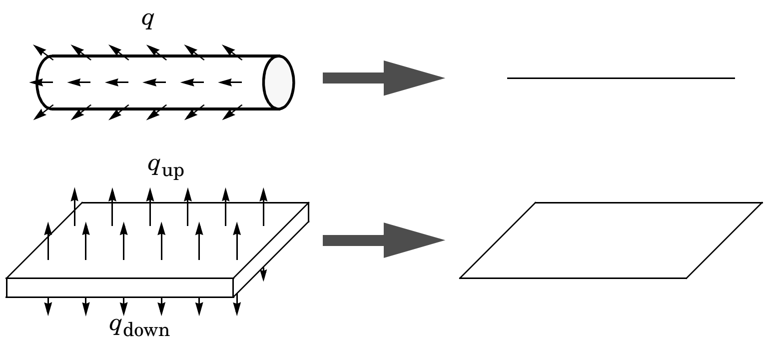


Figure 4-15: Geometry reduction from 3D to 1D (top) and from 3D to 2D (bottom).

The reduced geometry does not include all the boundaries of the original 3D geometry. For example, the reduced geometry does not represent the upside and downside surfaces of the plate in Figure 4-15 as boundaries.

Out-of-Plane Radiation and Out-of-Plane Heat Flux

Equation Formulation

2D GEOMETRIES

In 2D geometries, the temperature is assumed to be constant in the out-of-plane direction (z-direction with default spatial coordinate names). The equation for heat transfer in solids, Equation 4-14, and in fluids, Equation 4-16, are replaced by:

$$d_z \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = d_z Q + q_0 \quad (4\text{-}113)$$

$$d_z \rho C_p \frac{\partial T}{\partial t} + \rho C_p d_z \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = d_z Q + q_0 \quad (4\text{-}114)$$

Here d_z is the thickness of the domain in the out-of-plane direction. Here, the conductive heat flux, $\mathbf{q}$, becomes

$$\mathbf{q} = -d_z k \nabla T$$

1D AXISYMMETRIC GEOMETRIES

In 1D axisymmetric geometries, the temperature is assumed to be constant in the out-of-plane direction (z-direction with default spatial coordinate names) in addition to the axisymmetry (ϕ-coordinate with default spatial coordinate names). The equation for heat transfer in solids, Equation 6-9 is replaced by

$$(2\pi r d_z)\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = (2\pi r d_z) Q + q_0 \quad (4\text{-}115)$$

where d_z is the thickness of the domain in the z-direction. The equation for heat transfer in fluids, Equation 6-3, is replaced by

$$(2\pi r d_z)\rho C_p \frac{\partial T}{\partial t} + (2\pi r d_z)\rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = (2\pi r d_z) Q + q_0 \quad (4\text{-}116)$$

Here, the conductive heat flux, $\mathbf{q}$, becomes

$$\mathbf{q} = -(2\pi r d_z) k \nabla T$$

1D GEOMETRIES

In 1D geometries, the temperature is assumed to be constant in the radial direction. The equation for heat transfer in solids, Equation 6-9 is replaced by

$$A_\mathrm{c} \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = A_\mathrm{c} Q + q_0 \quad (4\text{-}117)$$

where A_c is the cross section of the domain in the plane perpendicular to the 1D geometry. The equation for heat transfer in fluids, Equation 6-3, is replaced by

$$A_\mathrm{c} \rho C_p \frac{\partial T}{\partial t} + A_\mathrm{c} \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = A_\mathrm{c} Q + q_0 \quad (4\text{-}118)$$

Here, the conductive heat flux, $\mathbf{q}$, becomes

$$\mathbf{q} = -A_{\mathrm{c}}k\nabla T$$

Out-of-plane flux conditions would apply to the exterior boundaries of the domain if the 1D geometry was seen as a cylinder. With the geometry reduction process, this heat flux condition is mathematically expressed using the cross section perimeter, P_{c}, as in:

$$q_0 = P_{\mathrm{c}}q_{0,\,z}$$

where $q_{0,\,z}$ is the heat flux density distributed along the cross section perimeter.

The Heat Transfer Coefficients

One of the most common boundary conditions when modeling heat transfer is convective cooling or heating whereby a fluid cools or heats a surface by natural or forced convection. In principle, it is possible to model this process in two ways:

- Using a heat transfer coefficient on the surfaces
- Extending the model to describe the flow and heat transfer in the surrounding fluid

The second approach is the most accurate if the geometry or the external flow is complicated. The Heat Transfer Module includes the Conjugate Heat Transfer predefined multiphysics coupling and the CFD Module includes the Non-Isothermal Flow predefined multiphysics coupling for this purpose. However, such a simulation can become costly, both in terms of computational time and memory requirement.

The first method is simple, yet powerful and efficient. The convective heat flux on the boundaries in contact with the fluid is then modeled as being proportional to the temperature difference across a fictitious thermal boundary layer. Mathematically, the heat flux is described by the equation

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\mathrm{ext}} - T)$$

where h is a heat transfer coefficient and T_{ext} the temperature of the external fluid far from the boundary.

The main difficulty in using heat transfer coefficients is in calculating or specifying the appropriate value of the h coefficient. That coefficient depends on the fluid's material properties, and the surface temperature — and, for forced convection, also on the fluid's flow rate. In addition, the geometrical configuration affects the coefficient. The Heat Transfer interface has built-in functions for the heat transfer coefficients. For most engineering purposes, the use of such coefficients is an accurate and numerically efficient modeling approach.

In this section:

- Defining the Heat Transfer Coefficients
- Nature of the Flow — The Grashof Number
- Heat Transfer Coefficients — External Natural Convection
- Heat Transfer Coefficients — Internal Natural Convection

- Heat Transfer Coefficients — External Forced Convection
- Heat Transfer Coefficients — Internal Forced Convection

Defining the Heat Transfer Coefficients

It is possible to divide the convective heat flux into four main categories depending on the type of convection condition (natural or forced) and on the type of geometry (internal or external flow). In addition, these cases can all experience either laminar or turbulent flow conditions, resulting in eight types of convection, as in Figure 4-16.

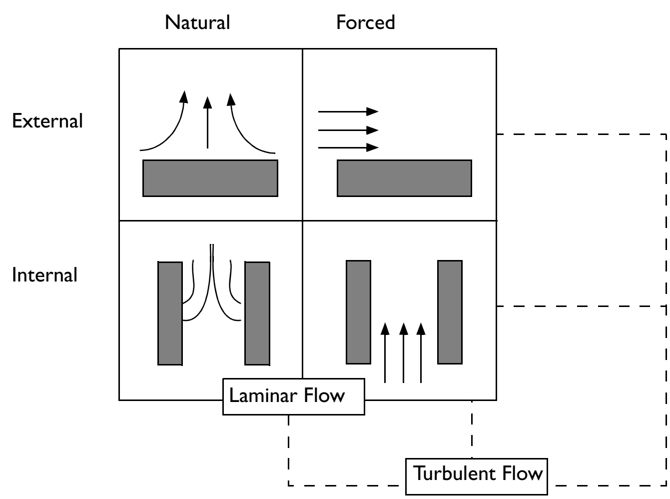


Figure 4-16: The eight categories of convective heat flux.

The difference between natural and forced convection is that in the forced convection an external force such as a fan creates the flow. In natural convection, buoyancy forces induced by temperature differences together with the thermal expansion of the fluid drive the flow.

Heat transfer books generally contain a large set of empirical and theoretical correlations for h coefficients. This module includes a subset of them. The expressions are based on the following set of dimensionless numbers:

- The Nusselt number, $\mathrm{Nu}_L = hL/k$
- The Reynolds number, $\mathrm{Re}_L = \rho UL/\mu$

- The Prandtl number, $\mathrm{Pr} = \mu C_p / k$
- The Rayleigh number, $\mathrm{Ra}_L = \mathrm{Gr}_L \mathrm{Pr} = \rho^2 g \alpha_p C_p \Delta T L^3 / (\mu k)$

where

- h is the heat transfer coefficient (SI unit: W/(m²·K))
- L is the characteristic length (SI unit: m)
- ΔT is the temperature difference between the surface and the external fluid bulk (SI unit: K)
- g is the acceleration of gravity (SI unit: m/s²)
- k is the thermal conductivity of the fluid (SI unit: W/(m·K))
- ρ is the fluid density (SI unit: kg/m³)
- U is the bulk velocity (SI unit: m/s)
- μ is the dynamic viscosity (SI unit: Pa·s)
- C_p is the heat capacity at constant pressure of the fluid (SI unit: J/(kg·K))
- α_p is the coefficient of thermal expansion (SI unit: 1/K)

Further, Gr_L refers to the *Grashof number,* which is the squared ratio of the viscous time scale to the buoyancy time scale multiplied by the Reynolds number.

Nature of the Flow — The Grashof Number

In cases of externally driven flow, such as forced convection, the nature of the flow is characterized by the Reynolds number, Re, which describes the ratio of the inertial forces to the viscous forces. However, the velocity scale is initially unknown for internally driven flows such as natural convection. In such cases the Grashof number, Gr, characterizes the flow. It describes the ratio of the time scales for viscous diffusion in the fluid and the internal driving force (the buoyancy force). Like the Reynolds number it requires the definition of a length scale, the fluid's physical properties, and the temperature scale (temperature difference). The Grashof number is defined as:

$$\mathrm{Gr}_L = \frac{g \alpha_p (T_s - T_0) L^3}{(\mu / \rho)^2}$$

where g is the acceleration of gravity, α_p is the fluid's coefficient of thermal expansion, T_s denotes the temperature of the hot surface, T_0 equals the temperature of the surrounding air, L is the length scale, μ represents the fluid's dynamic viscosity, and ρ its density.

In general, the coefficient of thermal expansion α_p is given by

$$\alpha_p = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)_p$$

which for an ideal gas reduces to

$$\alpha_p = \frac{1}{T}$$

The transition from laminar to turbulent flow occurs at a Gr value of 10^9; the flow is turbulent for larger values.

The Rayleigh number, Ra, is another indicator of the regime. It is similar to the Grashof number except that it accounts for the thermal diffusivity: Ra = Pr Gr. A small value of the Ra number indicates that the conduction dominates. It such case using heat transfer coefficients to model convective heat transfer is not relevant. Instead, modeling the fluid as immobile is likely to be accurate.

Heat Transfer Coefficients — External Natural Convection

VERTICAL WALL

The following correlations correspond to equations 9.26 and 9.27 in Ref. 16:

$$h = \begin{cases} \dfrac{k}{L}\left(0.68 + \dfrac{0.67\mathrm{Ra}_L^{1/4}}{\left(1+\left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{4/9}}\right) & \text{if } \mathrm{Ra}_L \le 10^9 \\ \dfrac{k}{L}\left(0.825 + \dfrac{0.387\mathrm{Ra}_L^{1/6}}{\left(1+\left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{8/27}}\right)^2 & \text{if } \mathrm{Ra}_L > 10^9 \end{cases} \qquad (4\text{-}119)$$

where the height of the wall, L, is a correlation input and

$$\mathrm{Ra}_L = \frac{g\alpha_p\rho^2 C_p|T-T_{\mathrm{ext}}|L^3}{k\mu} \qquad (4\text{-}120)$$

where in turn g is the acceleration of gravity equal to 9.81 m/s^2. All material properties are evaluated at $(T+T_{\mathrm{ext}})/2$. This correlation is valid for $10^4 \le \mathrm{Ra}_L \le 10^{13}$.

INCLINED WALL

The following correlations correspond to equations 9.26 and 9.27 in Ref. 16 (the same as for a vertical wall):

$$h = \begin{cases} \dfrac{k}{L}\left(0.68 + \dfrac{0.67(\cos(\phi)\mathrm{Ra}_L)^{1/4}}{\left(1+\left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{4/9}}\right) & \text{if } \mathrm{Ra}_L \le 10^9 \\ \dfrac{k}{L}\left(0.825 + \dfrac{0.387\mathrm{Ra}_L^{1/6}}{\left(1+\left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{8/27}}\right)^2 & \text{if } \mathrm{Ra}_L > 10^9 \end{cases} \qquad (4\text{-}121)$$

where the length of the wall, L, is a correlation input and ϕ is the tilt angle (the angle between the wall and the vertical direction; $\phi = 0$ for vertical walls). These correlations are valid for $-60° < \phi < 60°$ and $10^4 \le \mathrm{Ra}_L \le 10^{13}$.

The definition of the Raleigh number, Ra_L, is analogous to the one for vertical walls and is given by the following:

$$\mathrm{Ra}_L = \frac{g\alpha_p\rho^2 C_p|T - T_{\mathrm{ext}}|L^3}{k\mu} \qquad (4\text{-}122)$$

where in turn g denotes the gravitational acceleration, equal to 9.81 m/s^2.

For turbulent flow, 1 is used instead of $\cos\phi$ in the expression for h, because this gives better accuracy (see Ref. 36).

According to Ref. 16, correlations for inclined walls are only satisfactory for the top side of a cold plate or the down face of a hot plate. Hence, these correlations are not recommended for the bottom side of a cold face and for the top side of a hot plate.

The laminar-turbulent transition depends on ϕ (see Ref. 36). Unfortunately, little data is available about transition. There is some data available in Ref. 36 but this data is only approximative, according to the authors. In addition, data is only provided for water (Pr around 6). For this reason, the flow is defined as turbulent, independently of the ϕ value, when

$$\frac{g\alpha_p\rho^2 C_p|T-T_{\text{ext}}|L^3}{k\mu} > 10^9$$

All material properties are evaluated at $(T+T_{\text{ext}})/2$.

HORIZONTAL PLATE, UPSIDE

The following correlations correspond to equations 9.30–9.32 in Ref. 16 but can also be found as equations 7.77 and 7.78 in Ref. 36.

If $T > T_{\text{ext}}$, then

$$h = \begin{cases} \frac{k}{L}0.54\text{Ra}_L^{1/4} & \text{if } 10^4 \le \text{Ra}_L \le 10^7 \\ \frac{k}{L}0.15\text{Ra}_L^{1/3} & \text{if } 10^7 \le \text{Ra}_L \le 10^{11} \end{cases} \tag{4-123}$$

while if $T \le T_{\text{ext}}$, then

$$h = \frac{k}{L}0.27\text{Ra}_L^{1/4} \text{ if } 10^5 \le \text{Ra}_L \le 10^{10} \tag{4-124}$$

Ra_L is given by Equation 4-120, and L, the plate diameter (defined as area/perimeter, see Ref. 36) is a correlation input. The material data are evaluated at $(T+T_{\text{ext}})/2$.

HORIZONTAL PLATE, DOWNSIDE

Equation 4-123 is used when $T \le T_{\text{ext}}$ and Equation 4-124 is used when $T > T_{\text{ext}}$. Otherwise it is the same implementation as for Horizontal Plate, Upside.

LONG HORIZONTAL CYLINDER

The following correlations correspond to equations 9.34 in Ref. 16. It is validated for $\text{Ra}_D \le 10^{12}$.

$$h = \frac{k}{D}\left(0{,}6 + \frac{0{,}387 Ra_D^{1/6}}{\left(1+\left(\frac{0{,}559}{Pr}\right)^{9/16}\right)^{8/27}}\right)^2 \tag{4-125}$$

Here D is the cylinder diameter and Ra_D is given by

$$\text{Ra}_D = \frac{g\alpha_p\rho^2 C_p|T-T_{\text{ext}}|D^3}{k\mu}$$

The material data are evaluated at $(T + T_{ext})/2$.

SPHERE

The following correlations correspond to equation 9.35 in Ref. 16. It is validated for $Ra_D \le 10^{11}$ and $Pr \ge 0.7$.

$$h = \frac{k}{D}\left(2 + \frac{0{,}589 Ra_D^{1/4}}{\left(1 + \left(\frac{0{,}469}{Pr}\right)^{9/16}\right)^{4/9}}\right)^2 \tag{4-126}$$

Here D is the cylinder diameter and Ra_D is given by

$$Ra_D = \frac{g\alpha_p\rho^2 C_p|T - T_{ext}|D^3}{k\mu}$$

The material data are evaluated at $(T + T_{ext})/2$.

VERTICAL THIN CYLINDER

The following correlation corresponds to equation 7.83 in Ref. 36. It is validated only for side walls of the thin cylinder ($\delta_T \ge D$), the horizontal disks (top and bottom) should be treated as horizontal plates. If the boundary thin layer is much smaller than D, vertical wall correlations should be used.

$$h = \frac{k}{H}\left(\frac{4}{3}\left(\frac{7Ra_H Pr}{5(20 + 21Pr)}\right)^{1/4} + \frac{4(272 + 315Pr)H}{35(64 + 63Pr)D}\right)$$

where D is the cylinder diameter, H is the cylinder height, and Ra_H is given by

$$Ra_H = \frac{g\alpha_p|T - T_{ext}|H^3}{k_\mu}$$

The material data are evaluated at $(T + T_{ext})/2$.

Heat Transfer Coefficients — Internal Natural Convection

NARROW CHIMNEY, PARALLEL PLATES

If $Ra_L < H/L$ and $T > T_{ext}$, then

$$h = \frac{k}{H}\frac{1}{24}Ra_L \tag{4-127}$$

where the plate distance, L, and the chimney height, H, are correlation inputs (equation 7.96 in Ref. 36). Ra_L is given by Equation 4-120. The material data are evaluated at $(T+T_{\mathrm{ext}})/2$.

NARROW CHIMNEY, CIRCULAR TUBE

If $\mathrm{Ra}_D < H/D$, then

$$h = \frac{k}{H}\frac{1}{128}\mathrm{Ra}_D$$

where the tube diameter, D, and the chimney height, H, are correlation inputs (table 7.2 in Ref. 36 with $D_{\mathrm{h}} = D$). Ra_D is given by Equation 4-120 with L replaced by D. The material data are evaluated at $(T+T_{\mathrm{ext}})/2$.

Heat Transfer Coefficients — External Forced Convection

PLATE, AVERAGED TRANSFER COEFFICIENT

This correlation is a combination of equations 7.34 and 7.41 in Ref. 16:

$$h = \begin{cases} 2\dfrac{k}{L}\dfrac{0.3387\mathrm{Pr}^{1/3}\mathrm{Re}_L^{1/2}}{(1+(0.0468/\mathrm{Pr})^{2/3})^{1/4}} & \text{if } \mathrm{Re}_L \le 5\cdot 10^5 \\ 2\dfrac{k}{L}\mathrm{Pr}^{1/3}(0.037\mathrm{Re}_L^{4/5}-871) & \text{if } \mathrm{Re}_L > 5\cdot 10^5 \end{cases} \tag{4-128}$$

where $\mathrm{Pr} = \mu C_p/k$ and $\mathrm{Re}_L = \rho U_{\mathrm{ext}}L/\mu$. The plate length, L, and the exterior velocity, U_{ext}, are correlation inputs. The material data are evaluated at $(T+T_{\mathrm{ext}})/2$.

PLATE, LOCAL TRANSFER COEFFICIENT

This correlation corresponds to equations 5.79b and 5.131 in Ref. 36:

$$h = \begin{cases} \dfrac{k}{\max(x, \sqrt{\mathrm{eps}})}0.332\mathrm{Pr}^{1/3}\mathrm{Re}_x^{1/2} & \text{if } \mathrm{Re}_x \le 5\cdot 10^5 \\ \dfrac{k}{\max(x, \sqrt{\mathrm{eps}})}0.0296\mathrm{Pr}^{1/3}\mathrm{Re}_x^{4/5} & \text{if } \mathrm{Re}_x > 5\cdot 10^5 \end{cases} \tag{4-129}$$

where $\mathrm{Pr} = \mu C_p/k$ and $\mathrm{Re}_x = \rho U_{\mathrm{ext}}x/\mu$. x, the position along the plate, and U_{ext}, the exterior velocity are correlation inputs. The material data are evaluated at $(T+T_{\mathrm{ext}})/2$.

ISOTHERMAL TUBE

This correlation corresponds to equations 8.55 and 8.61 in Ref. 16:

$$h = \begin{cases} \dfrac{k}{D}3.66 & \text{if } \mathrm{Re}_D \le 2500 \\ \dfrac{k}{D}0.027\mathrm{Re}_D^{4/5}\mathrm{Pr}^n\left(\dfrac{\mu}{\mu(T)}\right)^{0.14} & \text{if } \mathrm{Re}_D > 2500 \end{cases} \tag{4-130}$$

where $\mathrm{Pr} = \mu C_p/k$, $\mathrm{Re}_D = \rho U_{\mathrm{ext}} D/\mu$ and $n = 0.3$ if $T < T_{\mathrm{ext}}$ and $n = 0.4$ if $T \ge T_{\mathrm{ext}}$. D (the tube diameter) and U_{ext} (the exterior velocity) are correlation inputs. All material data are evaluated at T_{ext} except $\mu(T)$, which is evaluated at the wall temperature, T.

Equivalent Thermal Conductivity Correlations

The Nusselt number **Nu** is the ratio of total heat flux to conductive heat flux. If the fluid flow is not solved, the heat equation can still use an equivalent conductivity to account for the convective heat flux in the conductive part. The conductivity is increased according to **Nu** to account for the contribution of the convective heat flux.

	Fluid node

Correlations giving **Nu** from various material properties for two configurations of rectangular enclosures are described below.

Horizontal Cavity With Bottom Heating

The following correlation corresponds to equation 9.49 in Ref. 16. It is validated for $3 \cdot 10^5 \le \mathrm{Ra}_H \le 7 \cdot 10^9$:

$$\mathrm{Nu} = 0.069\mathrm{Ra}_H^{1/3}\mathrm{Pr}^{0.074} \tag{4-131}$$

where Ra_H is computed from the height H of the cavity and the temperatures T_1 and T_2 of the bottom and top walls, and $\mathrm{Pr} = \mu C_p / k$. The material data are evaluated at $(T_1 + T_2)/2$.

Vertical Cavity With Sidewall Heating

The following correlations correspond to equations 9.50, 9.51, and 9.52 in Ref. 16:

- If $1 \le \dfrac{H}{L} \le 2$, $10^{-3} \le \mathrm{Pr} \le 10^5$, and $10^3 \le \dfrac{\mathrm{Pr}\mathrm{Ra}_L}{0.2+\mathrm{Pr}}$:

$$\mathrm{Nu} = 0.18\left(\frac{\mathrm{Pr}}{0.2+\mathrm{Pr}}\mathrm{Ra}_L\right)^{0.29} \tag{4-132}$$

- If $2 \le \dfrac{H}{L} \le 10$, $\mathrm{Pr} \le 10^5$, and $10^3 \le \mathrm{Ra}_L \le 10^{10}$:

$$\mathrm{Nu} = 0.22\left(\frac{\mathrm{Pr}}{0.2+\mathrm{Pr}}\mathrm{Ra}_L\right)^{0.28}\left(\frac{H}{L}\right)^{-1/4} \tag{4-133}$$

- If $10 \le \frac{H}{L} \le 40$, $1 \le \mathrm{Pr} \le 2 \cdot 10^4$, and $10^4 \le \mathrm{Ra}_L \le 10^7$:

$$\mathrm{Nu} = 0.42\mathrm{Ra}_L^{1/4}\mathrm{Pr}^{0.012}\left(\frac{H}{L}\right)^{-0.3} \tag{4-134}$$

where H is the height of the cavity, L is the distance between the side plates, Ra_L is computed from L and the temperatures T_1 and T_2 of the side walls, and $\mathrm{Pr} = \mu C_p/k$. The material data are evaluated at $(T_1 + T_2)/2$.

Temperature Dependence of Surface Tension

The variation of the surface tension σ (SI unit: N/m) with temperature at fluid interfaces must be taken into account for the computation of phenomena such as Marangoni effect. These temperature dependencies are available in the form of a coefficient library for a set of liquid-gas interfaces.

For a liquid water-air interface, the following quadratic relation is used:

$$\sigma = -2.3519705 \cdot 10^{-7} T^2 - 1.63350014 \cdot 10^{-5} T + 9.77001279 \cdot 10^{-2}$$

where T (SI unit: K) is the temperature.

In other cases, a linear relation is used instead:

$$\sigma = A(T - T_0) + B$$

where T_0 is a reference temperature taken at 0 °C (273.15 K) and the coefficients A (SI unit: N/(m·K)) and B (SI unit: N/m) are given in the following table for some liquid-gas interfaces:

TABLE 4-4: COEFFICIENTS OF SURFACE TENSION

INTERFACE	A	B	REFERENCE
Acetone - Air	$-1.120 \cdot 10^{-4}$	$2.626 \cdot 10^{-2}$	Ref. 37
Acetic Acid - Air	$-0.994 \cdot 10^{-4}$	$2.958 \cdot 10^{-2}$	Ref. 37
Ethanol - Air	$-0.832 \cdot 10^{-4}$	$2.405 \cdot 10^{-2}$	Ref. 37
Toluene - Air	$-1.189 \cdot 10^{-4}$	$3.09 \cdot 10^{-2}$	Ref. 37
Diethyl Ether - Air	$-0.908 \cdot 10^{-4}$	$1.892 \cdot 10^{-2}$	Ref. 37
Glycerol - Air	$-0.885 \cdot 10^{-4}$	$6.517 \cdot 10^{-2}$	Ref. 37
Heptane - Nitrogen	$-0.980 \cdot 10^{-4}$	$2.21 \cdot 10^{-2}$	Ref. 38
Mercury - Mercury (Vapor)	$-2.049 \cdot 10^{-4}$	$49.06 \cdot 10^{-2}$	Ref. 38
Ethylene Glycol - Ethylene Glycol (Vapor)	$-0.890 \cdot 10^{-4}$	$5.021 \cdot 10^{-2}$	Ref. 38

More data can be found in Ref. 37 and Ref. 38.

Heat Flux and Heat Balance

The concept of heat flux is not as simple as it first might seem. The reason is that heat is not a conserved quantity. Instead, the conserved quantity is the total energy. Hence, there is both a heat flux and an energy flux that are similar but not identical.

This section briefly describes the theory for the variables for Total Heat Flux and Energy Flux, used when computing Heat and Energy Balance. The definitions of these postprocessing variables do not affect the computational results, only variables available for results analysis and visualization.

In this section:

- Total Heat Flux and Energy Flux
- Heat and Energy Balance

Total Heat Flux and Energy Flux

TOTAL HEAT FLUX

The total heat flux vector is defined as (Ref. 5):

$$\mathbf{q}_{\mathrm{tot}} = \rho\mathbf{u}E + \mathbf{q} + \mathbf{q}_{\mathrm{r}} \tag{4-135}$$

where E is the internal energy. It is the sum of *convective* heat flux, $\rho\mathbf{u}E$, *conductive* heat flux, $\mathbf{q}$, and *radiative* heat flux, $\mathbf{q}_{\mathrm{r}}$. Hence, the total heat flux accounts for all three kinds of heat transfer described in The Physical Mechanisms under Heat Transfer. Recall that the internal energy is related to the enthalpy, H, via the following for a fluid (see Thermodynamic Description of Heat Transfer):

$$E = H - \frac{p}{\rho}$$

or the following for a solid:

$$E = H + \mathbf{P}:\mathbf{F}$$

The total heat flux vector, $\mathbf{q}_{\mathrm{tot}}$, is more suited to check the heat balance as described in the next section Heat and Energy Balance.

TOTAL ENERGY FLUX

The total energy flux is equal to:

$$\mathbf{e}_{\mathrm{tot}} = \rho\mathbf{u}\left(\frac{1}{2}\mathbf{u}\cdot\mathbf{u}\right) + \rho\mathbf{u}(E+\Psi) + \mathbf{q} + \mathbf{q}_{\mathrm{r}} - \sigma\mathbf{u}$$

Again, *convective* heat flux, $\rho\mathbf{u}E$, *conductive* heat flux, $\mathbf{q}$, and *radiative* heat flux, $\mathbf{q}_{\mathrm{r}}$ are accounted in the sum. The additional terms that complete the total energy flux are the convected kinetic energy, $\rho\mathbf{u}(\mathbf{u}\cdot\mathbf{u}/2)$, force potential energy, $\rho\mathbf{u}\Psi$, and stress, $-\sigma\mathbf{u}$. For a fluid, this expression becomes (Ref. 5, chapter 3.5):

$$\mathbf{e}_{\mathrm{tot}} = \rho\mathbf{u}\left(\frac{1}{2}\mathbf{u}\cdot\mathbf{u}\right) + \rho\mathbf{u}(E+\Psi) + \mathbf{q} + \mathbf{q}_{\mathrm{r}} - (-p\mathbf{I}+\tau)\mathbf{u}$$

Introducing the total internal energy, E_0, and total enthalpy, H_0:

$$E_0 = E + \frac{1}{2}\mathbf{u}\cdot\mathbf{u} \qquad H_0 = H + \frac{1}{2}\mathbf{u}\cdot\mathbf{u}$$

leads to the following equivalent expression involving total enthalpy:

$$\mathbf{e}_{\mathrm{tot}} = \rho\mathbf{u}(H_0+\Psi) + \mathbf{q} + \mathbf{q}_{\mathrm{r}} - \tau\mathbf{u}$$

The potential Ψ has a simple form in some special cases — for example, for gravitational effects (Chapter 1.4 in Ref. 35) — but it is in general rather difficult to derive. Potential energy is therefore often excluded and the total energy flux is approximated by

$$\mathbf{e}_{\mathrm{tot}} = \rho\mathbf{u}E_0 + \mathbf{q} + \mathbf{q}_{\mathrm{r}} - \sigma\mathbf{u} \tag{4-136}$$

The total energy flux vector, $\mathbf{e}_{\mathrm{tot}}$, is more suited to check the energy balance as described in the next section Heat and Energy Balance.

Heat and Energy Balance

HEAT BALANCE

This section assumes a heat transfer model that only solves for the temperature T. In particular, for a fluid, the velocity field $\mathbf{u}$ and pressure field p are user defined or computed from another physics interface. In this case, the heat balance in a domain follows the identity below (chapter 11.2 in Ref. 5), derived from Equation 4-11. It expresses the idea that internal energy variations in time and net heat flux are balanced by external heat and work sources.

$$\frac{d}{dt}\int_{\Omega} \rho E dv + \int_{\partial\Omega_{\mathrm{ext}}} \mathbf{q}_{\mathrm{tot}} \cdot \mathbf{n} ds = Q_{\mathrm{Int}} - W_{\mathrm{diss}} \tag{4-137}$$

The different variables in this formula are defined in Total Heat Flux and Energy Flux. For this equality to be true, the provided dependent variables (velocity field $\mathbf{u}$ and pressure field p for the Navier-Stokes equations) must satisfy a mass and a momentum conservation equation. The dissipation power, W_{diss}, contains both pressure work and viscous dissipation in fluids. The heat sources Q_{Int} include domain sources, interior boundary, edge and point sources, and radiative source at interior boundaries.

In 2D and 3D components if isolated point or edge source is not adjacent to a boundary, these are not included in Q_{Int}. In this case, these need to be computed separately.

Equation 4-137 is more visually represented by the diagram of Figure 4-17 below.

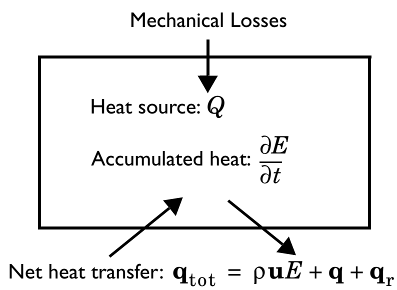


Figure 4-17: A heat balance diagram.

Several predefined variables are available in COMSOL Multiphysics to describe the heat rates involved in the system heat balance. See Global Variables for their definition.

ENERGY BALANCE

When the temperature T is solved together with additional mass and momentum equations, the total energy flux also becomes a conserved quantity and the following equation holds (chapter 11.1 in Ref. 5):

$$\frac{d}{dt}\int_{\Omega} \rho E_0 dv + \int_{\partial\Omega_{\mathrm{ext}}} \mathbf{e}_{\mathrm{tot}} \cdot \mathbf{n} ds = Q_{\mathrm{Int}} \tag{4-138}$$

The variables in this formula are defined in Total Heat Flux and Energy Flux. Equation 4-138 is more visually represented by the diagram of Figure 4-18 below.

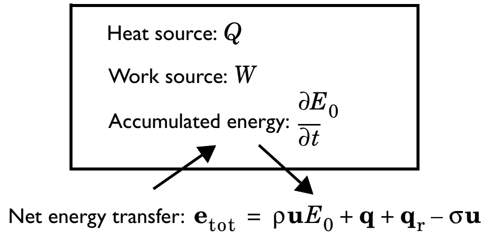


Figure 4-18: An energy balance diagram.

Several predefined variables are available in COMSOL Multiphysics to describe the energy rates involved in the system energy balance. See Global Variables for their definition.

Frames for the Heat Transfer Equations

In this section:

- Material and Spatial Frames
- Conversion Between Material and Spatial Frames

Material and Spatial Frames

The heat equation can be formulated either in a *spatial* coordinate system, with coordinate axes fixed in space, or in a *material* coordinate system, fixed to the material in its reference configuration and following the material as it deforms. COMSOL Multiphysics refers to these coordinate systems as the *spatial frame* and the *material frame*. In the case of immobile and undeformed materials, these two frames coincide.

USE OF FRAMES

The *spatial frame* is well adapted to simulate heat transfer in liquids and gases, where it is unreasonable to follow the state of individual material particles. The temperature is computed at fixed positions in space.

In solids, the *material frame* is more convenient. The temperature is computed at material particles uniquely identified by their position in some given reference configuration. It makes in particular the anisotropic material properties (thermal conductivity for example) independent of the current spatial orientation of the material.

In the heat transfer interfaces, the variables and equations are all defined on the spatial frame, and depending on the features, the user inputs may be defined on the material or spatial frame. Hence, they must be internally converted into the spatial frame if some deformation occurs.

POSITION VECTORS AND DEFORMATION GRADIENT

The position vector in the physical space is identified by the lowercase symbol $\mathbf{x}$ and lowercase letters x, y, and z for each coordinate (or r, φ, and z in axisymmetric components). After a given transformation, the position of an elementary volume is modified in the spatial frame but not in the material frame. The position vector in the this material frame is denoted by the uppercase symbol $\mathbf{X}$ and uppercase letters X, Y, and Z for each coordinate (or R, Φ, and Z in axisymmetric components).

The relation between $\mathbf{x}$ and $\mathbf{X}$ is carried by the *deformation gradient*:

$$F = \begin{bmatrix} \frac{\partial x}{\partial X} & \frac{\partial x}{\partial Y} & \frac{\partial x}{\partial Z} \\ \frac{\partial y}{\partial X} & \frac{\partial y}{\partial Y} & \frac{\partial y}{\partial Z} \\ \frac{\partial z}{\partial X} & \frac{\partial z}{\partial Y} & \frac{\partial z}{\partial Z} \end{bmatrix} \tag{4-139}$$

It relates elementary distances $d\mathbf{x}$ and $d\mathbf{X}$ in the domain, expressed in material and spatial frames, according to:

$$d\mathbf{x} = F d\mathbf{X} \tag{4-140}$$

The determinant of the deformation gradient, $\det(F)$, is the *volume ratio* field. In COMSOL Multiphysics, $\det(F)$ should always be strictly positive. Otherwise, the negative value is likely to be caused by an inverted mesh during the resolution of the model since it corresponds to a mathematical reflection operation.

The deformation gradient tensor and its determinant are essential in the conversion of physical quantities presented in the next paragraphs between material and spatial frames.

Note: In COMSOL Multiphysics, the variables `spatial.F11`, `spatial.F12`, …, store the coefficient of the transpose of the deformation gradient tensor F.

- About Frames in the *COMSOL Multiphysics Reference Manual.*
- Handling Frames in Heat Transfer

Conversion Between Material and Spatial Frames

This section explains how the user inputs are converted between material and spatial frames. The conversion depends on the dimension of the variables (scalars, vectors, or tensors) and on the density order.

As described in the previous paragraph Material and Spatial Frames, lowercase letters are used to denote the spatial frame coordinates while uppercase letters denote the material frame coordinates. In the followings, a physical quantity A will be referred to as $A_{(x,\,y,\,z)}$ in the spatial frame and to as $A_{(X,\,Y,\,Z)}$ in the material frame.

The equations solved by the heat transfer interfaces are written in the spatial frame. When an input is specified in the material frame, conversion is necessary to deduce $A_{(x,\,y,\,z)}$ from $A_{(X,\,Y,\,Z)}$.

DENSITY, HEAT SOURCE, HEAT FLUX

In heat transfer, the following variables are relative scalars of weight one (also called scalar densities):

- Mass density, ρ (SI unit: $\mathrm{kg/m^3}$),
- Heat source, Q_0 (SI unit: $\mathrm{W/m^3}$),
- Production/absorption coefficient, q_s (SI unit: $\mathrm{W/(m^3{\cdot}K)}$),
- Heat flux, q_0 (SI unit: $\mathrm{W/m^2}$),
- Heat transfer coefficient, h (SI unit: $\mathrm{W/(m^2{\cdot}K)}$).

For all these variables, the conversion between material and spatial frame follows the relation:

$$A_{(x,\,y,\,z)} = \frac{A_{(X,\,Y,\,Z)}}{\det(F)}$$

This way, the integral of volumetric quantities over the domain, such as the mass density, is invariant between frames:

$$\int_{\Omega_0} \rho_{(X,\,Y,\,Z)} d\tau_0 = \int_{\Omega_0} \rho_{(x,\,y,\,z)} \det(F) d\tau_0 = \int_{\Omega} \rho_{(x,\,y,\,z)} d\tau$$

In these equalities, Ω_0 and Ω denote the same domain but represented in material or in spatial frame, respectively. As expected, the same mass is found by integrating $\rho_{(X,\,Y,\,Z)}$ over the domain in the material frame or by integrating $\rho_{(x,\,y,\,z)}$ over the domain in the spatial frame. The same invariance principle applies to quantities per unit area, in particular heat flux and heat transfer coefficient:

$$\int_{\partial\Omega_0} h_{(X,\,Y,\,Z)} ds_0 = \int_{\partial\Omega_0} h_{(x,\,y,\,z)} \det(F) ds_0 = \int_{\partial\Omega} h_{(x,\,y,\,z)} ds$$

Here, $\partial\Omega_0$ and $\partial\Omega$ are the boundaries of the same domain in material and spatial frames, respectively.

VELOCITY

The relationship between the velocity vectors in material and spatial frames, $\mathbf{u}_{(X, Y, Z)}$ and $\mathbf{u}_{(x, y, z)}$, is

$$\mathbf{u}_{(x, y, z)} = F\mathbf{u}_{(X, Y, Z)}$$

This is directly deduced from the differential relation of Equation 4-140.

THERMAL CONDUCTIVITY

Thermal conductivity, k, is a tensor density. The relationship between the value on the spatial frame and the material frame is:

$$k_{(x, y, z)} = \frac{1}{\det(F)}Fk_{(X, Y, Z)}F^{\mathrm{T}}$$

With this relation, and recalling that

$$\mathbf{n}_{(X, Y, Z)} = F^{\mathrm{T}}\mathbf{n}_{(x, y, z)}$$

$$\nabla_{(X, Y, Z)}T = F^{\mathrm{T}}\nabla_{(x, y, z)}T$$

the total conductive heat flux through a boundary, computed in both frames according to the integrals below, gives the same result:

$$\int_{\partial\Omega_0} -k_{(X, Y, Z)}\nabla_{(X, Y, Z)}T \cdot \mathbf{n}_{(X, Y, Z)}ds_0 = \int_{\partial\Omega} -k_{(x, y, z)}\nabla_{(x, y, z)}T \cdot \mathbf{n}_{(x, yz)}ds$$

Here, $\partial\Omega_0$ and $\partial\Omega$ are the boundaries of the same domain in material and spatial frames, respectively.

THERMAL CONDUCTIVITY OF A LAYER

The same transformations are applied to thermal conductivity but with different transformation matrices. The deformation gradient tensor depends on the layer type:

- When the layer is resistive, the deformation gradient tensor F_{xdim} is equal to the deformation gradient tensor F defined in Equation 4-139.
- When the layer is conductive, the deformation gradient tensor F_{t} is defined using tangential derivatives as follows:

$$F_{\mathbf{t}} = \begin{bmatrix} xT_X & xT_Y & xT_Z \\ yT_X & yT_Y & yT_Z \\ zT_X & zT_Y & zT_Z \end{bmatrix}$$

where xT_X corresponds to the tangential derivative x with respect to X, and so on.

- When the layer is an extra dimension, the deformation gradient tensor F_{xdim} is defined as follows:

$$F_{\mathrm{xdim}} = \begin{bmatrix} xT_X + n_x n_X & xT_Y + n_x n_Y & xT_Z + n_x n_Z \\ yT_X + n_y n_X & yT_Y + n_y n_Y & yT_Z + n_y n_Z \\ zT_X + n_z n_X & zT_Y + n_z n_Y & zT_Z + n_z n_Z \end{bmatrix}$$

where xT_X corresponds to the tangential derivative x with respect to X, and so on. The (n_x, n_y, n_z) vector corresponds to the normal vector in the spatial frame, and the (n_X, n_Y, n_Z) vector corresponds to the normal vector in the material frame.

TIME DERIVATIVE

Partial differential equations often involve time derivative of a physical quantity such as temperature or internal energy in heat transfer. The variations of such state variables during an elementary time step are studied for a same elementary volume that could be subjected to spatial transformations. The material derivative, denoted d/dt, is the derivation operator used in such cases. The following relation defines the material derivative in the spatial frame.

$$\frac{d}{dt} = \frac{\partial}{\partial t} + \mathbf{u}_{(x,\,y,\,z)} \cdot \nabla_{(x,\,y,\,z)}$$

The right-hand side of this relation shows a new term $\mathbf{u} \cdot \nabla$ corresponding to convection in the case of fluids, or convected quantity by translational motion of a solid.

- About Frames in the *COMSOL Multiphysics Reference Manual*.
- Handling Frames in Heat Transfer

References

1. G.A. Holzapfel, *Nonlinear Solid Mechanics*, John Wiley & Sons, 2000.

2. T. Poinsot and D. Veynante, *Theoretical and Numerical Combustion*, 2nd ed., Edwards, 2005.

3. J.M. Powers, “On the Necessity of Positive Semi-Definite Conductivity and Onsager Reciprocity in Modeling Heat Conduction in Anisotropic Media,” *ASME J. Heat Transfer*, vol. 126, pp. 670–675, 2004.

4. G.A. Maugin, *The Thermomechanics of Nonlinear Irreversible Behaviors: An Introduction*, World Scientific, 1999.

5. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, 2nd ed., John Wiley & Sons, 2007.

6. S. Jacques, S. Rastegar, S. Thomsen, and M. Motamedi, *Nonlinear Finite-element Analysis The Role of Dynamic Changes in Blood Perfusion and Optical Properties in Laser Coagulation of Tissue*, IEEE J. Selected Topics in Quantum Electronics, vol. 2, issue 4, pp. 922–933, 1996.

7. S. Bhowmick, J.E. Coad, D.J. Swanlund, J.C. Bischof, “In vitro thermal therapy of AT-1 Dunning prostate tumors” *Int. J. Hyperthermia*, vol. 20, no. 1, pp. 73–92, 2004.

8. F. Xu, K.A. Seffen and T.J. Lu, “Temperature-Dependent Mechanical Behaviors of Skin Tissue,” *IAENG Int. J. Computer Science*, vol. 35, no 1, 2008.

9. M. Pop, A. Molckovsky, L. Chin, M.C. Kolios, M.A. Jewett, M.D. Sherar, “Changes in dielectric properties at 460 kHz of kidney and fat during heating: importance for radio-frequency thermal therapy”, Phys. Med. Biol., vol. 48, 2003 (http://www.ncbi.nlm.nih.gov/pubmed/12953912/).

10. P.A. Hasgall, F. Di Gennaro, C. Baumgartner, E. Neufeld, M.C. Gosselin, D. Payne, A. Klingenböck, N. Kuster, *IT’IS Database for thermal and electromagnetic parameters of biological tissues*, Version 3.0, 2015. www.itis.ethz.ch/database

11. C. Rossmann and D. Haemmerich, *Review of Temperature Dependence of Thermal Properties, Dielectric Properties, and Perfusion of Biological Tissues at Hyperthermic and Ablation Temperatures*, Critical Reviews in Biomedical Engineering, Vol. 42, pp. 467-492, 2014.

12. D.A. Nield and A. Bejan, *Convection in Porous Media*, in Convection Heat Transfer, Fourth Edition, John Wiley & Sons, Inc., Hoboken, NJ, USA, 2013.

13. EN 15026, *Hygrothermal performance of building components and building elements - Assessment of moisture transfer by numerical simulation*, CEN, 2007.

14. J. Bear and Y. Bachmat, *Introduction to Modeling of Transport Phenomena in Porous Media*, Kluwer Academic Publisher, 1990.

15. R. Sieger and J. Howell, *Thermal Radiation Heat Transfer*, 4th ed., Taylor & Francis, New York, 2002.

16. F.P. Incropera, D.P. DeWitt, T.L. Bergman, and A.S. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th ed., John Wiley & Sons, 2006.

17. http://www.esrl.noaa.gov/gmd/grad/solcalc

18. M.F. Modest, *Radiative Heat Transfer*, 2nd ed., Academic Press, San Diego, California, 2003.

19. W.A. Fiveland, "The Selection of Discrete Ordinate Quadrature Sets for Anisotropic Scattering," *Fundamentals of Radiation Transfer*, HTD, vol. 160, ASME, 1991.

20. A. Bejan et al., *Heat Transfer Handbook*, John Wiley & Sons, 2003.

21. F. Charron, *Partage de la chaleur entre deux corps frottants*, Publication Scientifique et Technique du Ministère de l'Air, no. 182, 1943. (In French)

22. D.C. Wilcox, *Turbulence Modeling for CFD*, 2nd ed., DCW Industries, 1998.

23. J. Larsson, *Numerical Simulation of Turbulent Flows for Turbine Blade Heat Transfer*, Doctoral Thesis for the Degree of Doctor of Philosophy, Chalmers University of Technology, Sweden, 1998.

24. W.M. Kays, "Turbulent Prandtl Number — Where Are We?", *ASME J. Heat Transfer*, vol. 116, pp. 284–295, 1994.

25. B. Weigand, J.R. Ferguson, and M.E. Crawford, "An Extended Kays and Crawford Turbulent Prandtl Number Model," *Int. J. Heat and Mass Transfer*, vol. 40, no. 17, pp. 4191–4196, 1997.

26. D. Lacasse, È. Turgeon, and D. Pelletier, "On the Judicious Use of the k-ε Model, Wall Functions and Adaptivity," *Int. J. Thermal Sciences*, vol. 43, pp. 925–938, 2004.

27. D.A. Nield, “Effects of local thermal non-equilibrium in steady convective processes in a saturated porous medium: forced convection in a channel,” *J. Porous Media*, vol. 1, 1998, pp. 181–186.

28. W.J. Minkowycz et al., “On departure from local thermal equilibrium in porous media due to a rapidly changing heat source: the Sparrow number,” *Int. J. Heat Mass Transfer*, vol. 42, 1999, pp. 3373–3385.

29. A. Amiri and K. Vafai, “Transient analysis of incompressible flow through a packed bed,” *Int. J. Heat Mass Transfer*, vol 41, 1998, pp. 4259–4279.

30. R.G. Carbonell and S. Whitaker, “Heat and Mass Transfer in Porous Media,” *Fundamentals of Transport Phenomena in Porous Media*, J. Bear and M.Y. Corapcioglu, eds., Springer, 1984, pp. 121–198.

31. J.L. Monteith and M.H. Unsworth, *Principles of Environmental Physics,* Edward Arnold, London, 290 pp., 1990.

32. P.T. Tsilingiris, “Thermophysical and Transport Properties of Humid Air at Temperature Range Between 0 and 100 °C,” *Energy Conversion and Management*, vol. 49, no. 5, pp. 1098–1110, 2008.

33. J. Zhang, A. Gupta, and J. Bakera, “Effect of Relative Humidity on the Prediction of Natural Convection Heat Transfer Coefficients,” *Heat Transfer Engineering*, vol. 28, no. 4, pp. 335–342, 2007.

34. W. Wagner and H-J Kretzschmar, *International Steam Tables*, 2nd ed., Springer, 2008.

35. M. Kaviany, *Principles of Convective Heat Transfer*, 2nd ed., Springer, 2001.

36. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1993.

37. Tables of Physical & Chemical Constants (16th edition 1995). 2.2.5 Surface tension. Kaye & Laby Online. Version 1.0 (2005), www.kayelaby.npl.co.uk

38. J.J. Jasper, “The Surface Tension of Pure Liquid Compounds”, *J. Phys. Chem. Ref. Data*, vol. 1, pp. 841–1010, 1972.

39. G.K. Batchelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.

5

The Heat Transfer Module Interfaces

The Heat Transfer Module includes five physics interfaces used to compute the temperature field, the radiative intensity field, or the relative humidity field:

TABLE 5-1: THE HEAT TRANSFER MODULE INTERFACES

INTERFACE	TAG	VERSIONS
Heat Transfer	ht	Heat Transfer
		Heat Transfer in Solids
		Heat Transfer in Fluids
		Bioheat Transfer
		Heat Transfer in Porous Media
		Heat Transfer in Building Materials
		Heat Transfer with Surface-to-Surface Radiation
		Heat Transfer with Radiation in Participating Media
Heat Transfer in Thin Shells	htsh	Heat Transfer in Thin Shells
		Heat Transfer in Thin Films
		Heat Transfer in Fractures
Surface-to-Surface Radiation	rad	Surface-to-Surface Radiation

TABLE 5-1: THE HEAT TRANSFER MODULE INTERFACES

INTERFACE	TAG	VERSIONS
Radiation in Participating Media	rpm	Radiation in Participating Media
Moisture Transport	mt	Moisture Transport

See also The Heat Transfer in Pipes Interface (htp) in the *Pipe Flow Module User's Guide*.

In this chapter:

- About the Heat Transfer Interfaces
- The Heat Transfer in Solids Interface
- The Heat Transfer in Fluids Interface
- The Heat Transfer Interface
- The Heat Transfer in Porous Media Interface
- The Heat Transfer in Building Materials Interface
- The Bioheat Transfer Interface
- The Heat Transfer with Surface-to-Surface Radiation Interface
- The Heat Transfer with Radiation in Participating Media Interface
- The Heat Transfer in Thin Shells Interface
- The Heat Transfer in Thin Films Interface
- The Heat Transfer in Fractures Interface
- The Surface-To-Surface Radiation Interface
- The Radiation in Participating Media Interface
- The Moisture Transport Interface

See also the Multiphysics Interfaces.

About the Heat Transfer Interfaces

The Heat Transfer Module includes the following interfaces:

- Heat Transfer (with several versions)
- Heat Transfer in Thin Shells (with several versions)
- Surface-to-Surface Radiation
- Radiation in Participating Media
- Moisture Transport

They are used to compute the temperature field, the radiative intensity field, and the relative humidity field.

The multiphysics interfaces also compute other physical fields like velocity, pressure, or electromagnetic fields, depending on the available COMSOL products. See Multiphysics Interfaces.

The main dependent variable is the temperature, T.

The various kinds of Heat Transfer interfaces and the thermal multiphysics couplings can be used for modeling heat transfer by conduction, convection, or radiation, as well as conjugate heat transfer, evaporation, and electromagnetic heating.

Space Dimensions

The physics interfaces are available in 1D, 2D, and 3D and for axisymmetric components with cylindrical coordinates in 1D and 2D.

All the interfaces except Heat Transfer in Thin Shells apply in domains, with features available at each geometric level (volumes, surfaces, edges, and points).

Study Types

Stationary and time-dependent studies are available with the Heat Transfer interfaces.

You can consider a heat transfer problem as stationary if the temperature field is independent of time at each point. The system is said to be at thermal equilibrium. It happens when the conditions are independent of time or vary on a time scale large enough so that they can be approximated as constant. This type of study can be used as an initial step for a time-dependent analysis.

For other cases, use a time-dependent study instead.

Study and Study Step Types in the *COMSOL Multiphysics Reference Manual*

Versions of the Heat Transfer Physics Interface

The versions of the Heat Transfer physics interface (`ht`) are:

- The Heat Transfer in Solids Interface
- The Heat Transfer in Fluids Interface
- The Heat Transfer Interface
- The Bioheat Transfer Interface
- The Heat Transfer in Porous Media Interface
- The Heat Transfer in Building Materials Interface
- The Heat Transfer with Surface-to-Surface Radiation Interface
- The Heat Transfer with Radiation in Participating Media Interface

After selecting a version, default nodes are added under the main node, which then defines which version of the Heat Transfer interface is added. Depending on the version of the physics interface selected, the default nodes vary. For example:

- If **Heat Transfer in Solids** ( ) is selected, a **Heat Transfer in Solids (ht)** interface is added with a default **Solid** model.
- If **Heat Transfer in Fluids** ( ) is selected, a **Heat Transfer in Fluids (ht)** interface is added with a default **Fluid** model.
- The **Heat Transfer (ht)** interface is automatically added when a multiphysics interface under the **Conjugate Heat Transfer** branch is added. It contains two default models: **Solid** (enabled by default) and **Fluid** (empty selection by default).

Benefits of the Different Heat Transfer Interfaces

The benefit of the different versions of the Heat Transfer interface, with `ht` as the common default name (see Heat Transfer Variables), is that it is easy to add the default settings when selecting the physics interface. At any time, add a **Fluid** or **Solid** node from the **Physics** toolbar — the functionality is always available.

Depending on the available COMSOL products, physics interface options are also available from a Heat Transfer interface by selecting a specific check box under the **Physical Model** section (for surface-to-surface radiation, biological tissue, radiation in participating media, porous media, or isothermal domain). See Table 5-2 and Table 5-4 for a description of the interface options.

TABLE 5-2: THE HEAT TRANSFER (HT) INTERFACE OPTIONS

ICON	NAME	DEFAULT PHYSICAL MODEL
	Heat Transfer in Solids	No check box is selected.
	Heat Transfer in Fluids	No check box is selected.
	Heat Transfer with Surface-to-Surface Radiation (under the **Radiation** branch)	The Surface-to-surface radiation check box is selected (which enables the **Radiation Settings** section).
	Heat Transfer with Radiation in Participating Media (under the **Radiation** branch)	The Radiation in participating media check box is selected (which enables the **Participating Media Settings** section).
	Bioheat Transfer	The Heat transfer in biological tissue check box is selected (which enables the **Damage Integral Analysis Discretization** section).
	Heat Transfer in Porous Media	The Heat Transfer in Porous Media check box is selected (which enables the **Porous matrix model** list).
	Heat Transfer in Building Materials	The Heat Transfer in Porous Media check box is selected (which enables the **Porous matrix model** list).
	Heat Transfer	No check box is selected.

Also see The Heat Transfer in Pipes Interface in the *Pipe Flow Module User's Guide* for simulating heat transfer in pipe networks, including wall heat transfer to the surroundings.

Versions of the Heat Transfer in Thin Shells Physics Interface

The versions of the Heat Transfer in Thin Shells physics interface (`htsh`) are:

- The Heat Transfer in Thin Shells Interface
- The Heat Transfer in Thin Films Interface
- The Heat Transfer in Fractures Interface

After selecting a version, default nodes are added under the main node, which then defines which version of the Heat Transfer in Thin Shells interface is added. Depending on the version of the physics interface selected, the default nodes vary:

- If **Heat Transfer in Thin Shells** ( ) is selected, a **Heat Transfer in Thin Shells (htsh)** interface is added with a default **Thin Conductive Layer** model.
- If **Heat Transfer in Thin Films** ( ) is selected, a **Heat Transfer in Thin Films (htsh)** interface is added with a default **Thin Film** model.
- If **Heat Transfer in Fractures** ( ) is selected, a **Heat Transfer in Fractures (htsh)** interface is added with a default **Fracture** model.

Benefits of the Different Heat Transfer in Thin Shells Interfaces

The benefit of the different versions of the Heat Transfer in Thin Shells interface, with `htsh` as the common default name (see Heat Transfer Variables), is that it is easy to add the default settings when selecting the physics interface. At any time, add a **Thin Conductive Layer** node from the **Physics** toolbar — the functionality is always available.

Depending on the available COMSOL products, physics interface options are also available from a Heat Transfer in Thin Shells interface by selecting a specific check box under the **Physical Model** section (for surface-to-surface radiation and porous media). See Table 5-3 and Table 5-4 for a description of the interface options.

TABLE 5-3: THE HEAT TRANSFER IN THIN SHELLS (HTSH) INTERFACE OPTIONS

ICON	NAME	DEFAULT PHYSICAL MODEL
	Heat Transfer in Thin Shells	No check box is selected.

TABLE 5-3: THE HEAT TRANSFER IN THIN SHELLS (HTSH) INTERFACE OPTIONS

ICON	NAME	DEFAULT PHYSICAL MODEL
	Heat Transfer in Thin Films	No check box is selected.
	Heat Transfer in Fractures	The Heat Transfer in Porous Media check box is selected.

Then, additional physics options are provided with the other interfaces and multiphysics interfaces:

TABLE 5-4: ADDITIONAL HEAT TRANSFER PHYSICS OPTIONS

ICON	NAME	ID	DEFAULT PHYSICAL MODEL
	Laminar Flow (under the **Conjugate Heat Transfer** branch)	—	See Table 5-2.
	Turbulent Flow k-ε and Turbulent Flow, Low Re k-ε (under the **Conjugate Heat Transfer** branch)	—	See Table 5-2.
	Thermoelectric Effect	—	No check boxes are selected under **Physical Model**.
	Surface-to-Surface Radiation (under the **Radiation** branch)	rad	No **Physical Model** section, but the **Radiation Settings** section is automatically available by default.
	Radiation in Participating Media (under the **Radiation** branch)	rpm	No **Physical Model** section, but the **Participating Media Settings** section is automatically available by default.
	Joule Heating (under the **Electromagnetic Heating** branch)	—	No check boxes are selected under **Physical Model**.
	Local Thermal Non-Equilibrium	—	See Table 5-2.
	Heat and Moisture	—	See Table 5-2.

More turbulent flow interfaces are available under the **Conjugate Heat Transfer** branch with the CFD Module:

- Turbulent Flow, Algebraic yPlus
- Turbulent Flow, L-VEL
- Turbulent Flow, k-ω
- Turbulent Flow, SST

- Turbulent Flow, Spalart-Allmaras

See The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces in the *CFD Module User's Guide* for more details.

More interfaces are available under the **Electromagnetic Heating** branch with ACDC, WaveOptics, and RF Modules.

See The Laser Heating Interface in the *Wave Optics Module User's Guide*, The Induction Heating Interface in the *AC/DC Module User's Guide*, and The Microwave Heating Interface in the *RF Module User's Guide* for more details.

Settings for the Heat Transfer Interface

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is ht.

PHYSICAL MODEL

In 2D and 1D axisymmetric components, set the **Thickness** d_z, which is the thickness of the domain in the out-of-plane direction. The default value is 1 m.

In 1D components, set the **Cross sectional area** A_c and the **Cross sectional perimeter** P_c of the domain. Default values are 1 m^2 and 1 m, respectively.

Some check boxes are also present in this section with certain COMSOL products.

> For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Click to select any of the following check boxes to activate the versions of the ht interface as described in Benefits of the Different Heat Transfer Interfaces:

- Select the **Surface-to-surface radiation** check box to enable surface-to-surface radiation features as described in The Heat Transfer with Surface-to-Surface Radiation Interface.

- Select the **Radiation in participating media** check box to enable radiation in participating media features. See The Heat Transfer with Radiation in Participating Media Interface.
- Select the **Heat transfer in biological tissue** check box to enable the Biological Tissue feature.
- Selecting **Isothermal domain** provides support for isothermal domain modeling. See Isothermal Domain Interface and Isothermal Domain.
- Select the **Heat Transfer in Porous Media** check box to enable the **Porous Medium**, **Fracture**, and **Building Material** features. See The Heat Transfer in Porous Media Interface and The Heat Transfer in Building Materials Interface.

AMBIENT SETTINGS

In this section, you can define ambient variables to be available as inputs from several features: the temperature T_{amb}, the absolute pressure p_{amb}, the relative humidity ϕ_{amb}, the wind velocity v_{amb}, and the solar irradiance $I_{\mathrm{s,amb}}$.

See Heat Flux, Diffuse Surface, Temperature, Initial Values, or External Radiation Source for examples of features of the Heat Transfer interface that have ambient variables available as inputs.

See Initial Values, Moisture Content, or Moisture Flux for examples of features of the Moisture Transport interface that have ambient variables available as inputs.

Two options are available for the definition of the **Ambient data**:

- When **User defined** (the default) is selected, the **Ambient temperature** T_{amb}, the **Ambient absolute pressure** p_{amb}, the **Ambient relative humidity** ϕ_{amb}, the **Wind velocity** v_{amb}, and the **Ambient solar irradiance** $I_{\mathrm{s,amb}}$ should be specified directly.
- When **Meteorological data (ASHRAE 2013)** is selected, the ambient variables are computed from monthly and hourly averaged measurements, made over several years at weather stations worldwide. See Using Ambient Data for more information. Further settings for the choice of the location, time, and ambient conditions are needed; and additional input fields are displayed underneath.

Location

In this section you can set the location by choosing among more than 6000 weather stations worldwide. Two options are available for the selection of the **Weather station**:

- When **From list** is selected, additional lists display underneath for the selection of a **Region**, a **Country**, and a **Station**. The **Country** list is refreshed for each selection in the **Region** list, and the **Station** list is refreshed for each selection in the **Country** list.
- It is also possible to select a station **From reference** by entering the corresponding **World Meteorological Organization reference**, which is a 6-digit number.

A single country may be available for more than one region selection if it has stations spread over different regions. For example, United States of America is available in the **Country** list when either North America, Eurasia, or Oceania is selected in the **Region** list.

When a station is selected **From list**, its World Meteorological Organization (WMO) reference is displayed below the **Station** list. The WMO references can also be retrieved from maps offered by third parties like the one available as of this product release on http://ashrae-meteo.info/.

Time

The **Date** and **Local time** should be set by entering values or expressions in the **Day**, **Month**, **Hour**, **Minute**, and **Second** fields of the two tables.

If the **Specify year** check box is selected, a value or expression for the **Year** should also be set. As the data are given as averages over several past years, this input is only used for the detection of leap years, in order to interpolate the data over the months.

For temporal studies, these inputs define the starting time of the simulation. By default, the **Update time from solver** check box is selected, and the time is then automatically updated with the time from the solver to evaluate the variables by interpolation of the measured data. Unselect this check box to manually set the time update.

See Ambient data interpolation for more information about the interpolation of data over months and hours.

See Processing of ASHRAE data for more information about the data.

A time unit suitable for simulations over a day or a year may be set in the **Study Settings** section of the **Time Dependent** node, by using for example **h** for hour, **d** for day, or **a** for a year. See Using Units in the *COMSOL Multiphysics Reference Manual* for more details.

Ambient conditions

Based on the measured data, several conditions are available for the **Temperature**, the **Dew point temperature**, and the **Wind speed**. The formula for each condition is recalled in Table 5-5, Table 5-6, and Table 5-7. The **Average** conditions correspond to weighted means of the measured data, whereas the other conditions are obtained by applying standard or modified deviations (**Low**, **High**, and **User defined coefficient for deviation** conditions), user defined corrections, or wind correlations to the average conditions; or by taking the minimum or maximum of the measured data (**Lowest** and **Highest** conditions). More information about these definitions can be found in Ambient Variables and Conditions.

TABLE 5-5: TEMPERATURE CONDITIONS

CONDITION	DEFINITION
Average	$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle$
Low	$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle - \sigma_{T,\,\mathrm{station}}$
High	$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle + \sigma_{T,\,\mathrm{station}}$
Lowest	$T_{\mathrm{amb}} = \min(T_{\mathrm{station}})$
Highest	$T_{\mathrm{amb}} = \max(T_{\mathrm{station}})$
User defined coefficient for deviation	$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle + c_{\sigma} \cdot \sigma_{T,\,\mathrm{station}}$
User defined correction	$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle + \Delta T$
Heating wind correlation[1]	$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle + \Delta T_{\mathrm{wind}}$
Cooling wind correlation[1]	$T_{\mathrm{amb}} = \langle T_{\mathrm{station}} \rangle - \Delta T_{\mathrm{wind}}$

[1]These correlations are not related to the wind speed conditions described in Table 5-7.

TABLE 5-6: DEW POINT TEMPERATURE CONDITIONS

CONDITION	DEFINITION
Average	$DPT_{\mathrm{amb}} = \langle DPT_{\mathrm{station}} \rangle$
Low	$DPT_{\mathrm{amb}} = \langle DPT_{\mathrm{station}} \rangle - \sigma_{DPT,\,\mathrm{station}}$
High	$DPT_{\mathrm{amb}} = \langle DPT_{\mathrm{station}} \rangle + \sigma_{DPT,\,\mathrm{station}}$
Lowest	$DPT_{\mathrm{amb}} = \min(DPT_{\mathrm{station}})$
Highest	$DPT_{\mathrm{amb}} = \max(DPT_{\mathrm{station}})$

TABLE 5-7: WIND SPEED CONDITIONS

CONDITION	DEFINITION
Average	$v_{\mathrm{amb}} = \langle v_{\mathrm{station}} \rangle$
Low	$v_{\mathrm{amb}} = \langle v_{\mathrm{station}} \rangle - \sigma_{v,\,\mathrm{station}}$
High	$v_{\mathrm{amb}} = \langle v_{\mathrm{station}} \rangle + \sigma_{v,\,\mathrm{station}}$
Lowest	$v_{\mathrm{amb}} = \min(v_{\mathrm{station}})$
Highest	$v_{\mathrm{amb}} = \max(v_{\mathrm{station}})$

The conditions set for **Temperature** and **Dew point temperature** should be consistent in order to keep the temperature larger than the dew point temperature. However, all settings combinations are available, and the relative humidity is majored by 1 when necessary.

By default, the ambient solar irradiance variable $I_{\mathrm{s,amb}}$ is defined from the measured data of clear sky noon beam normal irradiance, which is the irradiance coming from the direct sun. Select the **Include horizontal diffuse solar irradiance** check box to include the clear sky noon irradiance provided by the rest of the sky. See Ambient Variables and Conditions for details

The **Clear sky noon beam normal irradiance** and the **Clear sky noon diffuse horizontal irradiance** are available through the post-processing variables `ht.Isn_amb` and `ht.Ish_amb`.

For an example of the use of user-defined ambient data, see:

Temperature Field in a Cooling Flange: Application Library path **Heat_Transfer_Module/Thermal_Processing/cooling_flange**

For an example of the use of meteorological ambient data, see:

Condensation Detection in an Electronic Device with Transport and Diffusion: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/condensation_electronic_device_transport_diffusion**

Condensation Detection in an Electronic Device: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/condensation_electronic_device**

CONSISTENT STABILIZATION

The **Streamline diffusion** check box is selected by default and should remain selected for optimal performance for heat transfer in fluids or other applications that include a convective or translational term. **Crosswind diffusion** provides extra diffusion in regions with sharp gradients. The added diffusion is orthogonal to the streamlines, so streamline diffusion and crosswind diffusion can be used simultaneously. The **Crosswind diffusion** check box is also selected by default.

INCONSISTENT STABILIZATION

The **Isotropic diffusion** check box is not selected by default.

Heat Transfer Consistent and Inconsistent Stabilization Methods

ADVANCED SETTINGS

Add both a **Heat Transfer (ht)** and a **Moving Mesh (ale)** interface (found under the **Mathematics>Deformed Mesh** branch when adding a physics interface) then click the **Show** button ( ) and select **Advanced Physics Options** to display this section.

When the component contains a moving mesh, the **Enable conversions between material and spatial frame** check box is selected by default. This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a moving mesh, and when this option is active, the heat transfer features automatically account for deformation effects on heat transfer properties. In particular the effects of volume changes on the density are considered. Rotation effects on the thermal conductivity of an anisotropic material and, more generally, deformation effects on an arbitrary thermal conductivity, are also covered. When the **Enable conversions between material and spatial frame** check box is not selected, the feature inputs (for example, Heat Source, Heat Flux, Boundary Heat Source, and Line Heat Source) are not converted and are instead defined on the **Spatial** frame.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. The shape functions used for the temperature are **Quadratic** for the modeling of heat transfer in solids, **Linear** for the modeling of heat transfer in fluids. See the description of each version of the physics interface for more details.

DEPENDENT VARIABLES

The Heat Transfer interfaces have the dependent variable **Temperature** T. The dependent variable names can be changed. Editing the name of a scalar dependent variable changes both its field name and the dependent variable name. If a new field name coincides with the name of another field of the same type, the fields share degrees of freedom and dependent variable names. A new field name must not coincide with the name of a field of another type or with a component name belonging to some other field.

Settings for the Heat Transfer in Thin Shells Interface

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is htsh.

SHELL THICKNESS

Define the **Shell thickness** d_s (SI unit: m) (see Equation 4-45). The default is 0.01 m.

OUT-OF-PLANE THICKNESS

For 2D components, define the **Out-of-plane thickness** d_z (SI unit: m) (see Equation 4-45). The default is 1 m.

PHYSICAL MODEL

Some check boxes are present in this section with certain COMSOL products.

	For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Click to select any of the following check boxes to activate related features:

- Select the **Surface-to-surface radiation** check box to add a **Radiation Settings** section. By selecting this check box you can also add an Opacity node.
- Select the **Heat Transfer in Porous Media** check box to enable the **Fracture** feature for the modeling of porous media. This check box is selected by default in The Heat Transfer in Fractures Interface.

AMBIENT SETTINGS

The settings are the same as for the Heat Transfer interface. See Ambient Settings for details.

RADIATION SETTINGS

To display this section for any version of the Heat Transfer in Thin Shells interface, select the **Surface-to-surface radiation** check box under the **Physical Model** section.

	See The Heat Transfer with Surface-to-Surface Radiation Interface for details about the **Surface-to-surface radiation method** and **Radiation resolution** settings.

Define the **Wavelength dependence of emissivity**.

- Keep the default value, **Constant,** to define a diffuse gray radiation model. In this case, the surface emissivity has the same definition for all wavelengths. The surface emissivity can still depend on other quantities, in particular on the temperature.
- For **Solar and ambient** define a diffuse spectral radiation model with two spectral bands, one for short wavelengths, $[0, \lambda_1]$, (solar radiation) and one for large wavelengths, $[\lambda_1, +\infty[$, (ambient radiation). It is then possible to define the **Intervals**

endpoint (SI unit: m), λ_1, to adjust the wavelength intervals corresponding to the solar and ambient radiation. The surface properties can then be defined for each spectral band. In particular it is possible to define the solar absorptivity for short wavelengths and the surface emissivity for large wavelengths.

- For **Multiple spectral bands** set the **Number of wavelength bands** value (**2** to **5**), to define a diffuse spectral radiation model. It is then possible to provide a definition of the surface emissivity for each spectral band. Update **Intervals endpoint** (SI unit: m), λ_1, λ_2, ..., to define the wavelength intervals $[\lambda_{i-1}, \lambda_i[$ for i from 1 to the **Number of wavelength bands**. Note that the first and the last endpoints, λ_0 and λ_N (with N equal to the value selected to define the **Number of wavelength bands**), are predefined and equal to 0 and $+\infty$ respectively.

Modify the **Transparent media refractive index** if it is different from 1 that corresponds to vacuum refractive index and that is usually a good approximation for air refractive index.

Also select the **Use radiation groups** check box to enable the ability of defining radiation groups, which can, in many cases, speed up the radiation calculations.

Select the **Surface-to-surface radiation method**: **Hemicube** (the default) or **Direct area integration**. See below for descriptions of each method.

- For **Direct area integration** select the **Radiation integration order**. Sharp angles and small gaps between surfaces can require a higher integration order for more accuracy but also more computational cost to evaluate the irradiation.
- For **Hemicube** select the **Radiation resolution** — **256** is the default.

Select **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic** to define the **Discretization level** used for the surface radiosity shape function.

CONSISTENT STABILIZATION

The **Streamline diffusion** check box is selected by default and should remain selected for optimal performance for heat transfer in fluids or other applications that include a convective or translational term. **Crosswind diffusion** provides extra diffusion in regions with sharp gradients. The added diffusion is orthogonal to the streamlines, so streamline diffusion and crosswind diffusion can be used simultaneously. The **Crosswind diffusion** check box is also selected by default.

INCONSISTENT STABILIZATION

The **Isotropic diffusion** check box is not selected by default.

Heat Transfer Consistent and Inconsistent Stabilization Methods

ADVANCED SETTINGS

Add both a **Heat Transfer in Thin Shells (htsh)** and a **Moving Mesh (ale)** interface (found under the **Mathematics>Deformed Mesh** branch when adding a physics interface) then click the **Show** button ( ) and select **Advanced Physics Options** to display this section.

When the component contains a moving mesh, the **Enable conversions between material and spatial frame** check box is selected by default. This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a moving mesh, and when this option is active, the heat transfer features automatically account for deformation effects on heat transfer properties. In particular the effects of volume changes on the density are considered. Rotation effects on the thermal conductivity of an anisotropic material and, more generally, deformation effects on an arbitrary thermal conductivity, are also covered. When the **Enable conversions between material and spatial frame** check box is not selected, the feature inputs (for example, Heat Source (Heat Transfer in Thin Shells Interface) and Heat Flux (Heat Transfer in Thin Shells Interface)) are not converted and are instead defined on the **Spatial** frame.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. The shape functions used for the temperature are **Quadratic** for the modeling of heat transfer in thin structures.

DEPENDENT VARIABLES

The Heat Transfer in Thin Shells interfaces have the dependent variable **Temperature** T. The dependent variable names can be changed. Editing the name of a scalar dependent variable changes both its field name and the dependent variable name. If a new field name coincides with the name of another field of the same type, the fields share degrees of freedom and dependent variable names. A new field name must not coincide with the name of a field of another type or with a component name belonging to some other field.

The Heat Transfer in Solids Interface

The **Heat Transfer in Solids** ( ) interface is used to model heat transfer in solids by conduction, convection, and radiation. A Solid model is active by default on all domains. All functionality for including other domain types, such as a fluid domain, is also available.

The temperature equation defined in solid domains corresponds to the differential form of the Fourier's law that may contain additional contributions like heat sources.

When this version of the physics interface is added, these default nodes are added to the **Model Builder**: **Solid**, **Thermal Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Heat Transfer is Solids** to select physics features from the context menu.

PHYSICAL MODEL

By default, no check boxes are selected under the **Physical Model** section.

DISCRETIZATION

By default, the shape functions used for the temperature are **Quadratic**.

See Settings for the Heat Transfer Interface for a description of the other settings.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Handling Frames in Heat Transfer
- Feature Nodes for the Heat Transfer in Solids Interface
- Theory for Heat Transfer in Solids

Feature Nodes for the Heat Transfer in Solids Interface

This section details the nodes available with The Heat Transfer in Solids Interface with default settings:

- Domain Nodes for the Heat Transfer in Solids Interface
- Boundary Nodes for the Heat Transfer in Solids Interface
- Edge Nodes for the Heat Transfer in Solids Interface
- Point Nodes for the Heat Transfer in Solids Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE HEAT TRANSFER IN SOLIDS INTERFACE

The Heat Transfer in Solids Interface has the following domain nodes:

- Change Cross Section
- Change Thickness
- Fluid
- Heat Source
- Initial Values
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Phase Change Material
- Pressure Work
- Solid
- Thermoelastic Damping
- Translational Motion
- Viscous Dissipation

When the **Isothermal domain** check box is selected in the **Physical Model** section, the Isothermal Domain node is also available from the context menu or the **Physics** toolbar **Domains** menu.

BOUNDARY NODES FOR THE HEAT TRANSFER IN SOLIDS INTERFACE

The Heat Transfer in Solids Interface has the following boundary nodes:

- Boundary Heat Source
- Continuity
- Deposited Beam Power
- External Temperature (Thin Layer)
- Heat Flux
- Heat Source (Thin Film)
- Inflow Heat Flux
- Layer Heat Source (Thin Layer)
- Line Heat Source on Axis
- Open Boundary
- Outflow
- Periodic Condition
- Symmetry
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film
- Thin Layer

When the **Isothermal domain** check box is selected in the **Physical Model** section, the Isothermal Domain Interface node is added by default and is also available from the context menu or the **Physics** toolbar **Boundaries** menu.

EDGE NODES FOR THE HEAT TRANSFER IN SOLIDS INTERFACE

The Heat Transfer in Solids Interface has the following edge nodes (3D components only):

- Line Heat Source (Thin Rod)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Thin Rod

POINT NODES FOR THE HEAT TRANSFER IN SOLIDS INTERFACE

The Heat Transfer in Solids Interface has the following point nodes:

- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Temperature (Thin Rod)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

Select the **Isothermal domain** check box to make the Isothermal Domain and Isothermal Domain Interface nodes available.

The Heat Transfer in Fluids Interface

The **Heat Transfer in Fluids** ( ) interface is used to model heat transfer in fluids by conduction, convection, and radiation. A Fluid model is active by default on all domains. All functionality for including other domain types, such as a solid domain, is also available.

The temperature equation defined in fluid domains corresponds to the convection-diffusion equation that may contain additional contributions like heat sources.

When this version of the physics interface is added, these default nodes are added to the **Model Builder**: **Fluid**, **Thermal Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Heat Transfer in Fluids** to select physics features from the context menu.

PHYSICAL MODEL

By default, no check boxes are selected under the **Physical Model** section.

DISCRETIZATION

By default, the shape functions used for the temperature are **Linear**.

The rest of the settings are the same as for The Heat Transfer in Solids Interface. See Settings for the Heat Transfer Interface for a description of the other settings.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

- Handling Frames in Heat Transfer
- Feature Nodes for the Heat Transfer in Fluids Interface
- Theory for Heat Transfer in Fluids

Feature Nodes for the Heat Transfer in Fluids Interface

This section details the nodes available with The Heat Transfer in Fluids Interface with default settings:

- Domain Nodes for the Heat Transfer in Fluids Interface
- Boundary Nodes for the Heat Transfer in Fluids Interface
- Edge Nodes for the Heat Transfer in Fluids Interface
- Point Nodes for the Heat Transfer in Fluids Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE HEAT TRANSFER IN FLUIDS INTERFACE

The Heat Transfer in Fluids Interface has the following domain nodes:

- Change Cross Section
- Change Thickness
- Fluid
- Heat Source
- Initial Values
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Phase Change Material
- Pressure Work
- Solid
- Thermoelastic Damping
- Translational Motion
- Viscous Dissipation

Boundary Nodes for the Heat Transfer in Fluids Interface

The Heat Transfer in Fluids Interface has the following boundary nodes:

- Boundary Heat Source
- Continuity
- Deposited Beam Power
- External Temperature (Thin Layer)
- Heat Flux
- Heat Source (Thin Film)
- Inflow Heat Flux
- Layer Heat Source (Thin Layer)
- Line Heat Source on Axis
- Open Boundary
- Outflow
- Periodic Condition
- Symmetry
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film
- Thin Layer

Edge Nodes for the Heat Transfer in Fluids Interface

The Heat Transfer in Fluids Interface has the following edge nodes (3D components only):

- Line Heat Source (Thin Rod)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Thin Rod

POINT NODES FOR THE HEAT TRANSFER IN FLUIDS INTERFACE

The Heat Transfer in Fluids Interface has the following point nodes:

- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Temperature (Thin Rod)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

Select the **Isothermal domain** check box to make the Isothermal Domain and Isothermal Domain Interface nodes available.

The Heat Transfer Interface

The **Heat Transfer** ( ) interface is automatically added when a predefined multiphysics interface under the **Conjugate Heat Transfer** branch is added.

It is used to model heat transfer in solids and fluids by conduction, convection, and radiation. A Solid model is active by default on all domains, and a Fluid model is also added but not active.

The settings and the feature nodes are the same as for The Heat Transfer in Fluids Interface.

The Heat Transfer in Porous Media Interface

The **Heat Transfer in Porous Media** interface ( ), found under the **Heat Transfer** branch ( ) when adding a physics interface, is used to model heat transfer by conduction, convection, and radiation in porous media. A **Porous Medium** model is active by default on all domains. All functionality for including other domain types, such as a solid domain, is also available.

The temperature equation defined in porous media domains corresponds to the convection-diffusion equation with thermodynamic properties averaging models to account for both solid matrix and fluid properties. This equation is valid when the temperatures into the porous matrix and the fluid are in equilibrium. If not, use The Local Thermal Non-Equilibrium Interface instead.

The physics interface is an extension of the generic Heat Transfer interface. When this physics interface is added, the following default nodes are added in the **Model Builder**: **Porous Medium**, **Thermal Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Heat Transfer in Porous Media** to select physics features from the context menu.

PHYSICAL MODEL

The capability to define material properties, boundary conditions, and more for porous media heat transfer is activated by selecting the **Heat Transfer in Porous Media** check box (see Figure 5-1).

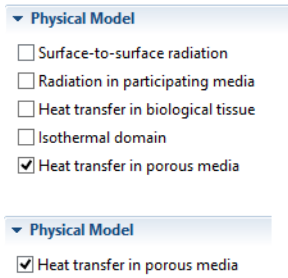


Figure 5-1: The capability to model porous media heat transfer is activated by selecting the Heat Transfer in Porous Media check box in any Settings window for Heat Transfer (ht) under Physical Model.

This check box is selected by default when adding The Heat Transfer in Porous Media Interface.

When the Subsurface Flow Module is added, under **Physical Model**, select **Extended** from the **Porous matrix model** list to use a version of the matrix feature to account for multiple immobile solids and fluids, as well as for geothermal heating.

Free Convection in Porous Media: Application Library path **Subsurface_Flow_Module/Heat_Transfer/convection_porous_medium**

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

DISCRETIZATION

By default, the shape functions used for the temperature are **Linear**.

The rest of the settings are the same as for The Heat Transfer in Solids Interface. See Settings for the Heat Transfer Interface for a description of the other settings.

- Feature Nodes for the Heat Transfer in Porous Media Interface
- Theory for Heat Transfer in Porous Media

Feature Nodes for the Heat Transfer in Porous Media Interface

This section details the nodes available with The Heat Transfer in Porous Media Interface with default settings:

- Domain Nodes for the Heat Transfer in Porous Media Interface
- Boundary Nodes for the Heat Transfer in Porous Media Interface
- Edge Nodes for the Heat Transfer in Porous Media Interface
- Point Nodes for the Heat Transfer in Porous Media Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE HEAT TRANSFER IN POROUS MEDIA INTERFACE

The Heat Transfer in Porous Media Interface has the following domain nodes:

- Building Material
- Change Cross Section
- Change Thickness
- Fluid
- Geothermal Heating
- Heat Source
- Immobile Fluids
- Initial Values
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Phase Change Material
- Porous Medium
- Pressure Work
- Solid
- Thermal Dispersion
- Thermoelastic Damping
- Translational Motion
- Viscous Dissipation

Set the **Porous matrix model** to **Extended** to make the Geothermal Heating and Immobile Fluids subnodes available.

Boundary Nodes for the Heat Transfer in Porous Media Interface

The Heat Transfer in Porous Media Interface has the following boundary nodes:

- Boundary Heat Source
- Continuity
- Deposited Beam Power
- External Temperature (Thin Layer)
- Fracture
- Heat Flux
- Heat Source (Thin Film)
- Inflow Heat Flux
- Layer Heat Source (Thin Layer)
- Line Heat Source on Axis
- Open Boundary
- Outflow
- Periodic Condition
- Symmetry
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film
- Thin Layer

Edge Nodes for the Heat Transfer in Porous Media Interface

The Heat Transfer in Porous Media Interface has the following edge nodes (for 3D components only):

- Line Heat Source (Thin Rod)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Thin Rod

POINT NODES FOR THE HEAT TRANSFER IN POROUS MEDIA INTERFACE

The Heat Transfer in Porous Media Interface has the following point nodes:

- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Temperature (Thin Rod)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

Select the **Isothermal domain** check box to make the Isothermal Domain and Isothermal Domain Interface nodes available.

The Heat Transfer in Building Materials Interface

The **Heat Transfer in Building Materials** ( ) interface is automatically added when the predefined multiphysics interface **Heat and Moisture Transport** is added. A Building Material model is active by default on all domains.

It is used to model heat transfer in building materials defined as porous media containing moisture, which is a mixture of liquid water and vapor. The temperature equation corresponds to the diffusion equation in which effective thermodynamic properties account for both the dry solid matrix and moisture properties. The latent heat of evaporation is included to define a heat source or sink.

When this physics interface is added, the following default nodes are added in the **Model Builder**: **Building Material**, **Thermal Insulation**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Heat Transfer in Building Materials** to select physics features from the context menu.

The settings and the feature nodes are the same as for The Heat Transfer in Porous Media Interface, except for the discretization of the temperature, for which **Quadratic** shape functions are used.

Theory for Heat Transfer in Building Materials

The Bioheat Transfer Interface

The **Bioheat Transfer** interface (), selected under the **Heat Transfer** branch ( ) when adding a physics interface, is used to model heat transfer by conduction, convection, and radiation. A **Biological Tissue** model is active by default on all domains. All functionality for including other domain types, such as a solid domain, are also available.

The temperature equation defined in biological tissue domains corresponds to the differential form of the Fourier's law with predefined contributions for bioheat sources. In addition, tissue damage integral models can be included, based on a temperature threshold or an energy absorption model.

When this version of the physics interface is added, these default nodes are added to the **Model Builder**: **Biological Tissue** (with a default Bioheat node), **Thermal Insulation** (the default boundary condition), and **Initial Values**. All functionality to include both solid and fluid domains are also available. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and sources. You can also right-click **Bioheat Transfer** to select physics features from the context menu.

PHYSICAL MODEL

The **Heat transfer in biological tissue** check box is selected by default, which enables the **Damage Integral Analysis Discretization** section.

DAMAGE INTEGRAL ANALYSIS DISCRETIZATION

Select the type and order of the shape function used for damaged tissue indicators. The default is **Discontinuous Lagrange** with **Constant** order.

The rest of the settings are the same as for The Heat Transfer in Solids Interface. See Settings for the Heat Transfer Interface for a description of the other settings.

- Theory for Bioheat Transfer
- Biological Tissue and Bioheat
- Feature Nodes for the Bioheat Transfer Interface

Hepatic Tumor Ablation: Application Library path **Heat_Transfer_Module/Medical_Technology/tumor_ablation**

Feature Nodes for the Bioheat Transfer Interface

This section details the nodes available with The Bioheat Transfer Interface with default settings:

- Domain Nodes for the Bioheat Transfer Interface
- Boundary Nodes for the Bioheat Transfer Interface
- Edge Nodes for the Bioheat Transfer Interface
- Point Nodes for the Bioheat Transfer Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE BIOHEAT TRANSFER INTERFACE

The Bioheat Transfer Interface has the following domain nodes:

- Bioheat
- Biological Tissue
- Change Cross Section
- Change Thickness
- Fluid
- Heat Source
- Initial Values
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Phase Change Material
- Pressure Work
- Solid
- Thermoelastic Damping
- Translational Motion
- Viscous Dissipation

BOUNDARY NODES FOR THE BIOHEAT TRANSFER INTERFACE

The Bioheat Transfer Interface has the following boundary nodes:

- Boundary Heat Source
- Continuity
- Deposited Beam Power
- External Temperature (Thin Layer)
- Heat Flux
- Heat Source (Thin Film)
- Inflow Heat Flux
- Layer Heat Source (Thin Layer)
- Line Heat Source on Axis
- Open Boundary
- Outflow
- Periodic Condition
- Symmetry
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film
- Thin Layer

EDGE NODES FOR THE BIOHEAT TRANSFER INTERFACE

The Bioheat Transfer Interface has the following edge nodes (for 3D components only):

- Line Heat Source (Thin Rod)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Thin Rod

POINT NODES FOR THE BIOHEAT TRANSFER INTERFACE

The Bioheat Transfer Interface has the following point nodes:

- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Temperature (Thin Rod)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

The Heat Transfer with Surface-to-Surface Radiation Interface

The **Heat Transfer with Surface-to-Surface Radiation (ht)** interface ( ), found under the **Radiation** branch ( ), is used to model heat transfer by conduction, convection, and radiation, including surface-to-surface radiation.

> Whereas The Surface-To-Surface Radiation Interface requires the temperature field as model input, this physics interface computes it. If the medium participates in the radiation (semi-transparent medium), then use The Heat Transfer with Radiation in Participating Media Interface instead.

A **Solid** model is active by default on all domains and when the **Surface-to-surface radiation** check box is selected for the Heat Transfer interface. All functionality to include other heat transfer models, such as **Fluid**, is also available. The radiosity equation defined on boundaries where surface-to-surface radiation is enabled corresponds to the radiosity method equation.

These default nodes are added to the **Model Builder**: **Solid** (with a default **Opacity** node, with **Opacity** set to **Transparent**), **Thermal Insulation**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Heat Transfer with Surface-to-Surface Radiation** to select physics features from the context menu.

PHYSICAL MODEL

When this physics interface is added, the **Surface-to-surface radiation** check box is selected by default, which enables the **Radiation Settings** section.

> See **Radiation Settings** for The Surface-To-Surface Radiation Interface. The rest of the settings are the same as for The Heat Transfer in Solids Interface. See Settings for the Heat Transfer Interface for a description of the other settings.

Thermo-Photo-Voltaic Cell: Application Library path **Heat_Transfer_Module/Thermal_Radiation/tpv_cell**

Feature Nodes for the Heat Transfer with Surface-to-Surface Radiation Interface

This section details the nodes available with The Heat Transfer with Surface-to-Surface Radiation Interface with default settings:

- Domain Nodes for the Heat Transfer with Surface-to-Surface Radiation Interface
- Boundary Nodes for the Heat Transfer with Surface-to-Surface Radiation Interface
- Edge Nodes for the Heat Transfer with Surface-to-Surface Radiation Interface
- Point Nodes for the Heat Transfer with Surface-to-Surface Radiation Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION INTERFACE

The Heat Transfer with Surface-to-Surface Radiation Interface has the following domain nodes:

- Change Cross Section
- Change Thickness
- Fluid
- Heat Source
- Initial Values
- Opacity
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Phase Change Material
- Pressure Work
- Solid
- Thermoelastic Damping
- Translational Motion
- Viscous Dissipation

Boundary Nodes for the Heat Transfer with Surface-to-Surface Radiation Interface

The Heat Transfer with Surface-to-Surface Radiation Interface has the following boundary nodes:

- Boundary Heat Source
- Continuity
- Deposited Beam Power
- Diffuse Mirror
- Diffuse Surface
- External Radiation Source
- External Temperature (Thin Layer)
- Heat Flux
- Heat Source (Thin Film)
- Inflow Heat Flux
- Layer Heat Source (Thin Layer)
- Line Heat Source on Axis
- Open Boundary
- Outflow
- Periodic Condition
- Prescribed Radiosity
- Radiation Group
- Symmetry
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film
- Thin Layer

Edge Nodes for the Heat Transfer with Surface-to-Surface Radiation Interface

The Heat Transfer with Surface-to-Surface Radiation Interface has the following edge nodes (for 3D components only):

- Line Heat Source (Thin Rod)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Thin Rod

POINT NODES FOR THE HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION INTERFACE

The Heat Transfer with Surface-to-Surface Radiation Interface has the following point nodes:

- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Temperature (Thin Rod)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

The Heat Transfer with Radiation in Participating Media Interface

The **Heat Transfer with Radiation in Participating Media (ht)** interface (), found under the **Heat Transfer>Radiation** branch (), is used to model heat transfer by conduction, convection, and radiation, including radiation in participating (semi-transparent) media.

Whereas The Radiation in Participating Media Interface requires the temperature field as model input, this physics interface computes it. If the medium does not participate in the radiation (transparent medium), then use The Heat Transfer with Surface-to-Surface Radiation Interface instead.

A **Solid** model is active by default on all domains and when the **Radiation in participating media** check box is selected for the Heat Transfer interface. All functionality to include other heat transfer models, like **Fluid**, is also available.

The radiative intensity equations defined in participating media domains are approximated either with the Rosseland approximation, P1 approximation, or discrete ordinates method.

The following default nodes are added to the **Model Builder**: **Solid**, **Initial Values**, **Thermal Insulation**, **Opaque Surface**, and **Continuity on Interior Boundary**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Heat Transfer with Radiation in Participating Media** to select physics features from the context menu.

PHYSICAL MODEL

When this physics interface is added, the **Radiation in participating media** check box is selected by default, which enables the **Participating Media Settings** section.

Except for the **Participating Media Settings**, the rest of the settings are the same as for The Heat Transfer in Solids Interface. See Settings for the Heat Transfer Interface for a description of the other settings.

PARTICIPATING MEDIA SETTINGS

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default), **Rosseland approximation**, or **P1 approximation**.

- When **Discrete ordinates method** is selected, Opaque Surface and Continuity on Interior Boundary are automatically added as default features.
- When **P1 approximation** is selected, Opaque Surface is automatically added as a default feature and both this and Incident Intensity are made available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or the context menu (all users). Continuity on Interior Boundary is not available.
- When **Rosseland approximation** is selected, neither Continuity on Interior Boundary nor Opaque Surface is included or available.

> The choice of **Radiation discretization method** also offers different settings for the Radiation in Participating Media (Heat Transfer Interface) (all methods), Opaque Surface (P1 approximation), and Incident Intensity (P1 approximation) nodes.

Refractive Index

For either selection, define the **Refractive index** n_{r} (dimensionless) of the participating media. The same refractive index is used for the whole model.

Performance Index

For **Discrete ordinates method** select a **Performance index** P_{index} from the list. Select a value between 0 and 1 that modifies the strategy used to define automatic solver settings. The default is 0.4. With small values, a robust setting for the solver is expected. With large values (up to 1), less memory is needed to solve the model.

Discrete Ordinates Method

For **Discrete ordinates method,** select an order from the list. This order defines the discretization of the radiative intensity direction.

> In 3D, S2, S4, S6, and S8 generate 8, 24, 48, and 80 directions, respectively. The default is S4.

In 2D, S2, S4, S6, and S8 generate 4, 12, 24, and 40 directions, respectively.

For additional background theory also see Discrete Ordinates Method (DOM), Discrete Ordinates Method Implementation in 2D, Rosseland Approximation Theory, and P1 Approximation Theory.

Discretization

For **Discrete ordinates method** or **P1 approximation** select the **Discretization level**: **Constant**, **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

Feature Nodes for the Heat Transfer with Radiation in Participating Media Interface

This section details the nodes available with The Heat Transfer with Radiation in Participating Media Interface with default settings:

- Domain Nodes for the Heat Transfer with Radiation in Participating Media Interface
- Boundary Nodes for the Heat Transfer with Radiation in Participating Media Interface
- Edge Nodes for the Heat Transfer with Radiation in Participating Media Interface
- Point Nodes for the Heat Transfer with Radiation in Participating Media Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Domain Nodes for the Heat Transfer with Radiation in Participating Media Interface

The The Heat Transfer with Radiation in Participating Media Interface has the following domain nodes:

- Change Cross Section
- Change Thickness
- Fluid
- Heat Source
- Initial Values
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Phase Change Material
- Pressure Work
- Radiation in Participating Media (Heat Transfer Interface)
- Solid
- Thermoelastic Damping
- Translational Motion
- Viscous Dissipation

Boundary Nodes for the Heat Transfer with Radiation in Participating Media Interface

The The Heat Transfer with Radiation in Participating Media Interface has the following boundary nodes:

- Boundary Heat Source
- Continuity
- Continuity on Interior Boundary
- Deposited Beam Power
- Diffuse Surface
- External Temperature (Thin Layer)
- Heat Flux
- Heat Source (Thin Film)
- Incident Intensity
- Inflow Heat Flux
- Layer Heat Source (Thin Layer)
- Line Heat Source on Axis
- Opaque Surface
- Open Boundary
- Outflow
- Periodic Condition
- Symmetry
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film
- Thin Layer

Edge Nodes for the Heat Transfer with Radiation in Participating Media Interface

The The Heat Transfer with Radiation in Participating Media Interface has the following edge nodes (for 3D components only):

- Line Heat Source (Thin Rod)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Thin Rod

Point Nodes for the Heat Transfer with Radiation in Participating Media Interface

The The Heat Transfer with Radiation in Participating Media Interface has the following point nodes:

- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Rod)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

The Heat Transfer in Thin Shells Interface

The **Heat Transfer in Thin Shells (htsh)** interface ( ), found in the **Thin Structures** physics area under the **Heat Transfer** branch ( ), is used to model heat transfer by conduction, convection and radiation in thin structures. A **Thin Conductive Layer** model is active by default on all boundaries. All functionalities for including other boundary contributions, such as surface-to-surface radiation, are also available.

The temperature equation defined on shells corresponds to the tangential differential form of the Fourier's law (see Equation 4-45) that may contain additional contributions such as heat sources.

The physics interface is available for 2D components, 3D components, and for axisymmetric components with cylindrical coordinates in 2D.

When this version of the physics interface is added, these default nodes are also added to the **Model Builder**: **Thin Conductive Layer**, **Insulation/Continuity** (a boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add additional nodes that implement, for example, edge or point conditions, and heat sources. You can also right-click **Heat Transfer in Thin Shells** to select physics features from the context menu.

PHYSICAL MODEL

By default, no check boxes are selected under the **Physical Model** section.

See Settings for the Heat Transfer in Thin Shells Interface for a description of the other settings.

- Handling Frames in Heat Transfer
- Feature Nodes for the Heat Transfer in Thin Shells Interface
- Theory for Heat Transfer in Thin Structures

Shell Conduction: Application Library path **Heat_Transfer_Module/Tutorials,_Thin_Structure/shell_conduction**

Feature Nodes for the Heat Transfer in Thin Shells Interface

This section details the nodes available with The Heat Transfer in Thin Shells Interface with default settings:

- Boundary Nodes for the Heat Transfer in Thin Shells Interface
- Edge Nodes for the Heat Transfer in Thin Shells Interface
- Point Nodes for the Heat Transfer in Thin Shells Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

The Heat Transfer in Thin Shells Interface does not have any domain node, as it can be applied only on boundaries.

BOUNDARY NODES FOR THE HEAT TRANSFER IN THIN SHELLS INTERFACE

The Heat Transfer in Thin Shells Interface has the following boundary nodes:

- Change Thickness (Heat Transfer in Thin Shells Interface)
- Continuity
- Deposited Beam Power
- Diffuse Surface
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Initial Values (Heat Transfer in Thin Shells Interface)
- Thin Conductive Layer (Heat Transfer in Thin Shells Interface)
- Thin Layered Shell (Heat Transfer in Thin Shells Interface)

EDGE NODES FOR THE HEAT TRANSFER IN THIN SHELLS INTERFACE

The Heat Transfer in Thin Shells Interface has the following edge nodes (for 3D components only):

- Change Effective Thickness (Heat Transfer in Thin Shells Interface)
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

POINT NODES FOR THE HEAT TRANSFER IN THIN SHELLS INTERFACE

The Heat Transfer in Thin Shells Interface has the following point nodes:

- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

The Heat Transfer in Thin Films Interface

The **Heat Transfer in Thin Films (htsh)** interface ( ), found in the **Thin Structures** physics area under the **Heat Transfer** branch ( ), is used to model heat transfer by conduction, convection and radiation in thin structures. A **Thin Film** model is active by default on all boundaries. All functionalities for including other boundary contributions, such as surface-to-surface radiation, are also available.

The temperature equation defined on thin films corresponds to the tangential differential form of the convection-diffusion equation that may contain additional contributions such as heat sources.

The physics interface is available for 2D components, 3D components, and for axisymmetric components with cylindrical coordinates in 2D.

When this version of the physics interface is added, these default nodes are also added to the **Model Builder**: **Thin Film**, **Insulation/Continuity** (a boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add additional nodes that implement, for example, edge or point conditions, and heat sources. You can also right-click **Heat Transfer in Thin Films** to select physics features from the context menu.

PHYSICAL MODEL

By default, no check boxes are selected under the **Physical Model** section.

See Settings for the Heat Transfer in Thin Shells Interface for a description of the other settings.

- Handling Frames in Heat Transfer
- Feature Nodes for the Heat Transfer in Thin Films Interface
- Theory for Heat Transfer in Thin Structures

Feature Nodes for the Heat Transfer in Thin Films Interface

This section details the nodes available with The Heat Transfer in Thin Films Interface with default settings:

- Boundary Nodes for the Heat Transfer in Thin Films Interface

- Edge Nodes for the Heat Transfer in Thin Films Interface
- Point Nodes for the Heat Transfer in Thin Films Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

The Heat Transfer in Thin Films Interface does not have any domain node, as it can be applied only on boundaries.

BOUNDARY NODES FOR THE HEAT TRANSFER IN THIN FILMS INTERFACE

The Heat Transfer in Thin Films Interface has the following boundary nodes:

- Change Thickness (Heat Transfer in Thin Shells Interface)
- Continuity
- Deposited Beam Power
- Diffuse Surface
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Initial Values (Heat Transfer in Thin Shells Interface)
- Thin Conductive Layer (Heat Transfer in Thin Shells Interface)
- Thin Film
- Thin Layered Shell (Heat Transfer in Thin Shells Interface)

EDGE NODES FOR THE HEAT TRANSFER IN THIN FILMS INTERFACE

The Heat Transfer in Thin Films Interface has the following edge nodes (for 3D components only):

- Change Effective Thickness (Heat Transfer in Thin Shells Interface)
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

POINT NODES FOR THE HEAT TRANSFER IN THIN FILMS INTERFACE

The Heat Transfer in Thin Films Interface has the following point nodes:

- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

The Heat Transfer in Fractures Interface

The **Heat Transfer in Fractures (htsh)** interface ( ), found in the **Thin Structures** physics area under the **Heat Transfer** branch ( ), is used to model heat transfer by conduction, convection and radiation in porous thin structures. A **Fracture** model is active by default on all boundaries. All functionalities for including other boundary contributions, such as surface-to-surface radiation, are also available.

The temperature equation defined on fractures corresponds to the tangential differential form of the convection-diffusion equation with thermodynamic properties averaging models to account for both solid matrix and fluid properties. The equation is valid when the temperatures into the porous matrix and the fluid are in equilibrium, and may contain additional contributions such as heat sources.

The physics interface is available for 2D components, 3D components, and for axisymmetric components with cylindrical coordinates in 2D.

When this version of the physics interface is added, these default nodes are also added to the **Model Builder**: **Fracture**, **Insulation/Continuity** (a boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add additional nodes that implement, for example, edge or point conditions, and heat sources. You can also right-click **Heat Transfer in Fractures** to select physics features from the context menu.

PHYSICAL MODEL

By default, the **Heat Transfer in Porous Media** check box is selected under the **Physical Model** section.

See Settings for the Heat Transfer in Thin Shells Interface for a description of the other settings.

- Handling Frames in Heat Transfer
- Feature Nodes for the Heat Transfer in Fractures Interface
- Theory for Heat Transfer in Thin Structures

Feature Nodes for the Heat Transfer in Fractures Interface

This section details the nodes available with The Heat Transfer in Fractures Interface with default settings:

- Boundary Nodes for the Heat Transfer in Fractures Interface
- Edge Nodes for the Heat Transfer in Fractures Interface
- Point Nodes for the Heat Transfer in Fractures Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

The Heat Transfer in Fractures Interface does not have any domain node, as it can be applied only on boundaries.

BOUNDARY NODES FOR THE HEAT TRANSFER IN FRACTURES INTERFACE

The Heat Transfer in Fractures Interface has the following boundary nodes:

- Change Thickness (Heat Transfer in Thin Shells Interface)
- Continuity
- Deposited Beam Power
- Diffuse Surface
- Fracture
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Initial Values (Heat Transfer in Thin Shells Interface)
- Thin Conductive Layer (Heat Transfer in Thin Shells Interface)
- Thin Layered Shell (Heat Transfer in Thin Shells Interface)

EDGE NODES FOR THE HEAT TRANSFER IN FRACTURES INTERFACE

The Heat Transfer in Fractures Interface has the following edge nodes (for 3D components only):

- Change Effective Thickness (Heat Transfer in Thin Shells Interface)
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

POINT NODES FOR THE HEAT TRANSFER IN FRACTURES INTERFACE

The Heat Transfer in Fractures Interface has the following point nodes:

- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

More nodes are available with more advanced settings. For the complete list of nodes available see Domain Features, Boundary Features, Edge Features, Point Features, and Global Features.

The Surface-To-Surface Radiation Interface

The **Surface-to-Surface Radiation (rad)** interface ( ), found under the **Heat Transfer>Radiation** branch ( ), is used to model heat transfer by radiation. It treats thermal radiation as an energy transfer between boundaries and external heat sources where the medium does not participate in the radiation (radiation in transparent media).

Whereas The Heat Transfer with Surface-to-Surface Radiation Interface computes the temperature field, this physics interface requires it as model input. If the medium participates in the radiation (semi-transparent medium), then use The Radiation in Participating Media Interface instead.

The radiosity equation defined on boundaries where surface-to-surface radiation is enabled corresponds to the radiosity method equation.

From the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Surface-to-Surface Radiation** to select physics features from the context menu. For the **Surface-to-Surface Radiation** interface, select a **Stationary** or **Time Dependent** study as a preset study type. The surface-to-surface radiation is always stationary (that is, the radiation time scale is assumed to be shorter than any other time scale), but the physics interface is compatible with all standard study types.

Absolute (thermodynamical) temperature units must be used. See Specifying Model Equation Settings in the *COMSOL Multiphysics Reference Manual*.

In this section:

- Settings for the Surface-to-Surface Radiation Interface
- Feature Nodes for the Surface-to-Surface Radiation Interface

Settings for the Surface-to-Surface Radiation Interface

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `rad`.

RADIATION SETTINGS

This section is always available for The Surface-To-Surface Radiation Interface. To display this section for any version of the Heat Transfer interface, select the **Surface-to-surface radiation** check box in the **Physical Model** section.

Define the **Wavelength dependence of emissivity**.

- Keep the default value, **Constant,** to define a diffuse gray radiation model. In this case, the surface emissivity has the same definition for all wavelengths. The surface emissivity can still depend on other quantities, in particular on the temperature.
- Select **Solar and ambient** to define a diffuse spectral radiation model with two spectral bands, one for short wavelengths, $[0, \lambda_1]$, (solar radiation) and one for large wavelengths, $[\lambda_1, +\infty[$, (ambient radiation). It is then possible to define the **Intervals endpoint** (SI unit: m), λ_1, to adjust the wavelength intervals corresponding to the solar and ambient radiation. The surface properties can then be defined for each spectral band. In particular it is possible to define the solar absorptivity for short wavelengths and the surface emissivity for large wavelengths.
- Select **Multiple spectral bands** and set the **Number of wavelength bands** value (**2** to **5**), to define a diffuse spectral radiation model. It is then possible to provide a definition of the surface emissivity for each spectral band. Update **Intervals endpoint** (SI unit: m), λ_1, λ_2, ..., to define the wavelength intervals $[\lambda_{i-1}, \lambda_i[$ for i from 1 to the **Number of wavelength bands**.

> The first and the last endpoints, λ_0 and λ_N (with N equal to the value selected to define the **Number of wavelength bands**), are predefined and equal to 0 and $+\infty$, respectively.

Modify the **Transparent media refractive index** if it is different from 1 and corresponds to vacuum refractive index, which is usually a good approximation for air refractive index.

In the **Exterior radiation** menu, select **Exterior is transparent** or **Exterior is opaque** to define if the exterior of the heat transfer interface selection should be considered as transparent or opaque, respectively, when **Opacity controlled** option is used to define **Radiation direction** in surface-to-surface radiation boundary conditions. It has no effect on a boundary where **Opacity controlled** option is not selected.

Also select the **Use radiation groups** check box to enable the ability to define radiation groups, which can, in many cases, speed up the radiation calculations.

Select the **Surface-to-surface radiation method**: **Hemicube** (the default) or **Direct area integration**.

- For Direct Area Integration select a **Radiation integration order** — **4** is the default.
- For Hemicube select a **Radiation resolution** — **256** is the default.

Select **Linear** (the default), **Quadratic**, **Cubic**, **Quartic** or **Quintic** to define the **Discretization level** used for the surface radiosity shape function.

Hemicube

Hemicube is the default method for the heat transfer interfaces. The more sophisticated and general hemicube method uses a z-buffered projection on the sides of a hemicube (with generalizations to 2D and 1D) to account for shadowing effects. Think of it as rendering digital images of the geometry in five different directions (in 3D; in 2D only three directions are needed), and counting the pixels in each mesh element to evaluate its view factor.

Its accuracy can be influenced by setting the **Radiation resolution** of the virtual snapshots. The number of z-buffer pixels on each side of the 3D hemicube equals the specified resolution squared. Thus the time required to evaluate the irradiation increases quadratically with resolution. In 2D, the number of z-buffer pixels is proportional to the resolution property, and thus the time is, as well.

For an axisymmetric geometry, G_{m} and F_{amb} must be evaluated in a corresponding 3D geometry obtained by revolving the 2D boundaries about the axis. COMSOL Multiphysics creates this virtual 3D geometry by revolving the 2D boundary mesh into a 3D mesh. The resolution can be controlled in the azimuthal direction by setting the number of azimuthal sectors, which is the same as the number of elements to a full revolution. Try to balance this number against the mesh resolution in the rz-plane.

Direct Area Integration

COMSOL Multiphysics evaluates the mutual irradiation between surface directly, without considering which face elements are obstructed by others. This means that shadowing effects (that is, surface elements being obstructed in nonconvex cases) are not taken into account. Elements facing away from each other are, however, excluded from the integrals.

Direct area integration is fast and accurate for simple geometries with no shadowing, or where the shadowing can be handled by manually assigning boundaries to different groups.

If shadowing is ignored, global energy is not conserved. Control the accuracy by specifying a **Radiation integration order**. Sharp angles and small gaps between surfaces may require a higher integration order for accuracy but also more time to evaluate the irradiation.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. This section is empty for The Surface-To-Surface Radiation Interface which defines the shape functions discretization in Radiation Settings.

- About the Heat Transfer Interfaces
- Feature Nodes for the Surface-to-Surface Radiation Interface
- The Surface-To-Surface Radiation Interface
- Theory for Surface-to-Surface Radiation

Feature Nodes for the Surface-to-Surface Radiation Interface

This section details the nodes available with The Surface-To-Surface Radiation Interface with default settings:

- Domain Nodes for the Surface-to-Surface Radiation Interface

- Boundary Nodes for the Surface-to-Surface Radiation Interface

	Some nodes are only available with some COMSOL products. For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Domain Nodes for the Surface-to-Surface Radiation Interface

The Surface-To-Surface Radiation Interface has one domain node: Opacity.

Boundary Nodes for the Surface-to-Surface Radiation Interface

The Surface-To-Surface Radiation Interface has the following boundary nodes:

- Continuity
- Diffuse Mirror
- Diffuse Surface
- External Radiation Source
- Prescribed Radiosity
- Radiation Group

The Radiation in Participating Media Interface

The **Radiation in Participating Media (rpm)** interface ( ), found under the **Heat Transfer>Radiation** branch ( ), is used to model radiative heat transfer inside participating (semi-transparent media) media.

Whereas The Heat Transfer with Radiation in Participating Media Interface computes the temperature field, this physics interface requires it as a model input. If the medium does not participate in the radiation (transparent medium), then use The Surface-To-Surface Radiation Interface instead.

The radiative intensity equations defined in participating media domains correspond to a P1 approximation or to the discrete ordinates method approximation equations depending on the selected approximation.

When the physics interface is added, these default nodes are added to the **Model Builder**: **Radiation in Participating Media**, **Wall**, **Continuity on Interior Boundary**, and **Initial Values**. Right-click the main node to add boundary conditions or other features. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Radiation in Participating Media** node to select physics features from the context menu.

In this section:

- Settings for the Radiation in Participating Media Interface
- Feature Nodes for the Radiation in Participating Media Interface

Settings for the Radiation in Participating Media Interface

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `rpm`.

- *Radiative Heat Transfer in Finite Cylindrical Media*: Application Library path **Heat_Transfer_Module/Verification_Examples/cylinder_participating_media**
- *Radiative Heat Transfer in a Utility Boiler*: Application Library path **Heat_Transfer_Module/Thermal_Radiation/boiler**

PARTICIPATING MEDIA SETTINGS

To display this section select the **Radiation in participating media** check box under **Physical Model** on any version of the **Settings** window for Heat Transfer.

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default) or **P1 approximation**.

- When **Discrete ordinates method** is selected, Opaque Surface and Continuity on Interior Boundary are automatically added as default features.
- When **P1 approximation** is selected, Opaque Surface is automatically added as a default feature and both this and Incident Intensity are made available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or the context menu (all users). Continuity on Interior Boundary is not available.

The choice of **Radiation discretization method** also offers different settings for the Radiation in Participating Media (Heat Transfer Interface) (all methods), Opaque Surface (P1 approximation), and Incident Intensity (P1 approximation) nodes.

Refractive Index

For either selection, define the **Refractive index** n_r (dimensionless) of the participating media. The same refractive index is used for the whole model.

Performance Index

When **Discrete ordinates method** is selected, choose a **Performance index** P_index from the list. Select a value between 0 and 1 that modifies the strategy used to define automatic solver settings. The default is 0.4. With small values, a robust setting for the solver is expected. With large values (up to 1), less memory is needed to solve the model.

Discrete Ordinates Method

When **Discrete ordinates method** is selected, choose the **Discrete ordinates method** order from the list. This order defines the discretization of the radiative intensity direction.

In 3D, S2, S4, S6, and S8 generate 8, 24, 48, and 80 directions, respectively. The default is S4.

In 2D, S2, S4, S6, and S8 generate 4, 12, 24, and 40 directions, respectively.

For additional background theory also see Discrete Ordinates Method (DOM), Discrete Ordinates Method Implementation in 2D, and P1 Approximation Theory.

Discretization

If **Discrete ordinates method** or **P1 approximation** is selected, select the **Discretization level**: **Constant**, **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

Feature Nodes for the Radiation in Participating Media Interface

This section details the nodes available with The Radiation in Participating Media Interface with default settings:

- Domain Nodes for the Radiation in Participating Media Interface
- Boundary Nodes for the Radiation in Participating Media Interface

Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE RADIATION IN PARTICIPATING MEDIA INTERFACE

The Radiation in Participating Media Interface has one domain node: Radiation in Participating Media (RPM Interface).

BOUNDARY NODES FOR THE RADIATION IN PARTICIPATING MEDIA INTERFACE

The Radiation in Participating Media Interface has the following boundary nodes:

- Continuity
- Continuity on Interior Boundary
- Incident Intensity
- Opaque Surface

The Moisture Transport Interface

The **Moisture Transport** ( ) interface is used to model moisture transfer in a porous medium. The underlying model provides an equation for the relative humidity in the medium that takes into account the moisture storage, the capillary suction forces, and the convective transport of vapor. A Porous Medium model is active by default on all domains.

When this physics interface is added, these default nodes are added to the **Model Builder**: **Porous Medium**, **Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Moisture Transport** to select physics features from the context menu.

Settings for the Moisture Transport Interface

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `mt`.

PHYSICAL MODEL

In 2D and 1D axisymmetric components, set the **Thickness** d_z, which is the thickness of the domain in the out-of-plane direction. The default value is 1 m.

In 1D components, set the **Cross sectional area** A_c and the **Cross sectional perimeter** P_c of the domain. Default values are 1 m^2 and 1 m, respectively.

CONSISTENT STABILIZATION

The **Streamline diffusion** check box is selected by default and should remain selected for optimal performance for applications that include a convective or translational term. **Crosswind diffusion** provides extra diffusion in regions with sharp gradients. The added diffusion is orthogonal to the streamlines, so streamline diffusion and crosswind diffusion can be used simultaneously. The **Crosswind diffusion** check box is also selected by default.

INCONSISTENT STABILIZATION

The **Isotropic diffusion** check box is not selected by default. To add isotropic diffusion, select the **Isotropic diffusion** check box. The field for the tuning parameter δ_{id} then becomes available. The default value is 0.25; increase or decrease the value of δ_{id} to increase or decrease the amount of isotropic diffusion.

In the *COMSOL Multiphysics Reference Manual:*

- Stabilization Techniques
- Stabilization

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. The shape functions used for the relative humidity are **Quadratic**.

In the *COMSOL Multiphysics Reference Manual* see Table 2-3 for links to common sections and Table 2-4 to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Theory for Moisture Transport

Feature Nodes for the Moisture Transport Interface

This section details the nodes available with The Moisture Transport Interface with default settings:

- Domain Nodes for the Moisture Transport Interface
- Boundary Nodes for the Moisture Transport Interface

Domain Nodes for the Moisture Transport Interface

The Moisture Transport Interface has the following domain nodes:

- Initial Values
- Moisture Source
- Porous Medium

Boundary Nodes for the Moisture Transport Interface

The Moisture Transport Interface has the following boundary nodes:

- Continuity
- Insulation
- Moisture Content
- Moisture Flux
- Symmetry
- Thin Moisture Barrier

6

The Heat Transfer Features

The Heat Transfer Interfaces have domain, boundary, edge, point, and pair nodes and subnodes (including out-of-plane and layer features) available. These nodes, listed in alphabetical order in this section, are available from the **Physics** ribbon toolbar (Windows users), from the **Physics** context menu (Mac or Linux users), or by right-clicking to access the context menu (all users). Subnodes are available by right-clicking the parent node and selecting it from the **Attributes** menu.

In this section:

Domain Features

The Heat Transfer interfaces have the following domain nodes and subnodes available:

- Bioheat
- Biological Tissue
- Building Material
- Change Cross Section
- Change Thickness
- Fluid
- Geothermal Heating
- Heat Source
- Immobile Fluids
- Initial Values
- Isothermal Domain
- Opacity
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Phase Change Material
- Porous Medium
- Pressure Work
- Radiation in Participating Media (Heat Transfer Interface)
- Radiation in Participating Media (RPM Interface)
- Solid
- Thermal Dispersion
- Thermoelastic Damping
- Translational Motion
- Viscous Dissipation

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Bioheat

This feature provides the source terms that represent blood perfusion and metabolism to model heat transfer in biological tissue using the bioheat equation:

$$\rho_{\mathrm{b}} C_{p,\,\mathrm{b}} \omega_{\mathrm{b}} (T_{\mathrm{b}} - T) + Q_{\mathrm{met}}$$

BIOHEAT

Enter values or expressions for these properties and source terms:

- **Arterial blood temperature** T_b, which is the temperature at which blood leaves the arterial blood veins and enters the capillaries. T is the temperature in the tissue, which is the dependent variable that is solved for and not a material property.
- **Specific heat, blood** $C_{p,\,b}$, which describes the amount of heat energy required to produce a unit temperature change in a unit mass of blood.
- **Blood perfusion rate** ω_b (SI unit: 1/s, which in this case means $(m^3/s)/m^3$), describes the volume of blood per second that flows through a unit volume of tissue.
- **Density, blood** ρ_b, which is the mass per volume of blood.
- **Metabolic heat source** Q_{met}, which describes heat generation from metabolism. Enter this quantity as the unit power per unit volume.

Hepatic Tumor Ablation: Application Library path
Heat_Transfer_Module/Medical_Technology/tumor_ablation

Microwave Heating of a Cancer Tumor: Application Library path
Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy

Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis: Application Library path
Heat_Transfer_Module/Medical_Technology/conical_dielectric_probe

LOCATION IN USER INTERFACE

A default **Bioheat** node is automatically added to the Biological Tissue node.

Context menus

Bioheat Transfer>Biological Tissue>Bioheat

More locations are available if the **Heat transfer in biological tissue** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Biological Tissue>Bioheat

Ribbon

Physics Tab with **Biological Tissue** selected in the model tree:

Attributes>Bioheat

Biological Tissue

This node adds the bioheat equation as the mathematical model for heat transfer in biological tissue. This equation can include source terms representing blood perfusion and metabolism using Pennes' approximation, through the addition of a **Bioheat** subnode; see Equation 4-19. Optionally it can define a damage model to account for overheating or freezing in tissues.

HEAT CONDUCTION, SOLID

The default **Thermal conductivity** k uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity and enter another value or expression in the field or matrix.

THERMODYNAMICS, SOLID

The default **Density** ρ and **Heat capacity at constant pressure** C_p are taken **From material**. The heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass. For **User defined** enter other values or expressions.

DAMAGED TISSUE

Select the **Include damage integral analysis** check box to account for overheating or freezing in tissues with a damage model. Two methods are available for the analysis; choose **Damage integral form**: **Temperature threshold** (the default) or **Energy absorption**. Depending to the material properties you have access to, you may choose one of the damage models.

The **Energy absorption** method is only applicable to hyperthermia analysis.

For **Temperature threshold**, define the settings for the Hyperthermia Analysis and Cryogenic Analysis. See First Form Integral for more details on the parameters of the model.

Hyperthermia Analysis

Enter values for:

- **Damage temperature** $T_{\mathrm{d,\,h}}$ to define the (high) temperature that the tissue needs to reach to start getting damaged. The default is 323.15 K.
- **Damage time** $t_{\mathrm{d,\,h}}$ to define the time needed for the necrosis to happen while the temperature is above $T_{\mathrm{d,\,h}}$. The default is 50 s.
- **Temperature of necrosis** $T_{\mathrm{n,\,h}}$ to define the (high) temperature to be reached for the necrosis to happen instantly. The default is 373.15 K.

Cryogenic Analysis

Enter values for:

- **Damage temperature** $T_{d, c}$ to define the (low) temperature that the tissue needs to reach to start getting damaged. The default is 273.15 K.
- **Damage time** $t_{d, c}$ to define the time needed for the necrosis to happen while the temperature is below $T_{d, c}$. The default is 50 s.
- **Temperature of necrosis** $T_{n, c}$ to define the (low) temperature to be reached for the necrosis to happen instantly. The default is 253.15 K.

For **Energy absorption**, define the Frequency Factor and Activation Energy to compute the degree of tissue injury with the Arrhenius equation. See Second Form Integral for more details.

Frequency Factor and Activation Energy

Enter values for:

- **Frequency factor** A in the Arrhenius equation. Default is taken **From material**. For **User defined** enter a value or an expression. The default is $7.39 \cdot 10^{39}\ \mathrm{s}^{-1}$.
- **Activation energy** ΔE in the Arrhenius equation. Default is taken **From material**. For **User defined** enter a value or an expression. The default is $2.577 \cdot 10^5$ J/mol.

As required, also define how to Use Different Material Properties for healthy and damaged tissue.

Use Different Material Properties

When the **Use different material properties for damaged tissue** check box is selected, choose a **Damaged material**, which can point to any material in the model. The default uses the **Domain material**. The healthy tissue properties correspond to the properties specified in the **Heat Conduction, Solid** and **Thermodynamics, Solid** sections. The effective tissue properties change from the healthy tissue properties to the damaged tissue properties as the damage evolves.

HEAT CONDUCTION, DAMAGED TISSUE

This section is available when the **Use different material properties for damaged tissue** check box is selected.

Select a **Thermal conductivity** k_d —**From material** (the default) or **User defined**, to be used for damaged tissue. For **User defined** choose **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity and enter another value or expression in the field or matrix.

THERMODYNAMICS, DAMAGED TISSUE

This section is available when the **Use different material properties for damaged tissue** check box is selected.

Select a **Density** ρ_d and **Heat capacity at constant pressure** $C_{p,\,d}$ —**From material** (the default) or **User defined**, to be used for damaged tissue. The heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass.

Theory for Bioheat Transfer

Damaged Tissue

When **Surface-to-surface radiation** is activated, the Opacity subnode is automatically added to the entire selection, with **Opaque** option selected. The domain selection can't be edited. To set some part of the domain as transparent, add a new Opacity subnode from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

There are specific predefined materials available in the Bioheat material database. See Materials Overview and Bioheat Material Database in the *COMSOL Multiphysics Reference Manual*.

Hepatic Tumor Ablation: Application Library path
Heat_Transfer_Module/Medical_Technology/tumor_ablation

Microwave Heating of a Cancer Tumor: Application Library path
Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy

Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis: Application Library path
Heat_Transfer_Module/Medical_Technology/conical_dielectric_probe

LOCATION IN USER INTERFACE

Context menus

Bioheat Transfer>Biological Tissue

More locations are available if the **Heat transfer in biological tissue** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Biological Tissue

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Bioheat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Biological Tissue**

Building Material

Use this node to model a building material as a porous medium containing a mixture of liquid water and vapor. The overall material properties change due to moisture transfer is accounted for through an apparent thermal conductivity change and a heat source or sink given in the divergence operator in function of the latent heat of evaporation:

$$(\rho C_p)_{\mathrm{eff}}\frac{\partial T}{\partial t} + \nabla\cdot\mathbf{q} = Q \tag{6-1}$$

$$\mathbf{q} = -(k_{\mathrm{eff}}\nabla T + L_{\mathrm{v}}\delta_{\mathrm{p}}\nabla(\phi p_{\mathrm{sat}})) \tag{6-2}$$

with the following material properties, fields, and source:

- $(\rho C_p)_{\mathrm{eff}}$ (SI unit: $\mathrm{J/(m^3{\cdot}K)}$) is the effective volumetric heat capacity at constant pressure.
- k_{eff} (SI unit: $\mathrm{W/(m{\cdot}K)}$) is the effective thermal conductivity (a scalar or a tensor if the thermal conductivity is anisotropic).
- L_{v} (SI unit: J/kg) is the latent heat of evaporation.
- δ_{p} (SI unit: s) is the vapor permeability.
- ϕ (dimensionless) is the relative humidity.
- p_{sat} (SI unit: Pa) is the vapor saturation pressure.
- Q (SI unit: $\mathrm{W/m^3}$) is the heat source (or sink). Add one or several heat sources as separate physics features. See Heat Source node for example.

For a steady-state problem the temperature does not change with time and the first term disappears.

MODEL INPUTS

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups are added, the model inputs appear here.

Relative humidity

This section has an input for the definition of the relative humidity, used in the right hand side of Equation 6-2.

The default **Relative humidity** ϕ is **User defined**. When additional physics interfaces are added to the model, the relative humidity variables defined by these physics interfaces can also be selected from the list. For example, if a **Moisture Transport** interface is added, you can select **Relative humidity (mt/pm1)** from the list.

If the node was added automatically after selecting the predefined multiphysics interface **Heat and Moisture Transport**, the relative humidity of the multiphysics node **Heat and Moisture** is used by default and the section is not editable. To edit the **Relative humidity** field, click **Make All Model Inputs Editable** ( ).

HEAT CONDUCTION

This section provides two options for the definition of the effective thermal conductivity k_{eff}:

- When **Equivalent thermal conductivity** is selected (the default), a value for the **Effective thermal conductivity** k_{eff} should be specified directly. The default **Effective thermal conductivity** is taken **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic** enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.
- When **Dry material thermal conductivity** is selected, the effective thermal conductivity is defined in function of the solid matrix and moisture properties:

$$k_{\text{eff}} = k_{\text{s}}\left(1+\frac{bw}{\rho_{\text{s}}}\right)$$

  This definition neglects the contribution due to the volume fraction change of the moist air.

  The **Dry solid thermal conductivity** k_{s} (SI unit: W/(m·K)) and the **Thermal conductivity supplement** b (dimensionless) should be specified. The default **Dry solid**

thermal conductivity and **Thermal conductivity supplement** are taken **From material**. For **User defined**, enter values or expressions into the editable fields.

The **Density** ρ_s and the **Moisture storage function** w are specified in the Thermodynamics, Dry Solid and Building Material Properties sections respectively.

THERMODYNAMICS, DRY SOLID

This section sets the thermodynamics properties of the dry solid.

The specific heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass of the dry solid material.

The **Density** ρ_s and the **Specific heat capacity** $C_{p,s}$ should be specified. The default **Density** and **Specific heat capacity** are taken **From material**. For **User defined**, enter values or expressions into the editable fields.

The effective volumetric heat capacity at constant pressure is defined to account for both solid matrix and moisture properties:

$$(\rho C_p)_{\mathrm{eff}} = \rho_s C_{p,\,s} + w C_{p,\,w}$$

where

- ρ_s (SI unit: kg/m^3) is the dry solid density.
- $C_{p,s}$ (SI unit: $J/(kg\cdot K)$) is the dry solid specific heat capacity.
- w (SI unit: kg/m^3) is the water content given by a moisture storage function.
- $C_{p,w}$ (SI unit: $J/(kg\cdot K)$) is the water heat capacity at constant pressure.

BUILDING MATERIAL PROPERTIES

This section sets the properties of the building material for moisture storage and vapor diffusion.

The **Moisture storage function** w should be set to characterize the relationship between the amount of accumulated water and the relative humidity in the material. The default **Moisture storage function** is taken **From material**. For **User defined**, enter another value or expression.

Two options are available for the specification of the building material properties for vapor diffusion:

- **Vapor permeability** (default) to define directly the vapor permeability δ_p. The default **Vapor permeability** is taken **From material**. For **User defined**, enter another value or expression.
- **Vapor resistance factor** μ to define the vapor permeability δ_p as:

$$\delta_p = \frac{\delta}{\mu}$$

where δ (SI unit: s) is the vapor permeability of still air. The default **Vapor resistance factor** is taken **From material**. For **User defined**, enter another value or expression.

If the node was added automatically after selecting the predefined multiphysics interface **Heat and Moisture Transport**, the building material properties of the multiphysics node **Heat and Moisture** are used by default and the inputs are not editable. To edit these fields, click **Make All Model Inputs Editable** ( ) in the **Model Inputs** section.

The Heat Transfer in Building Materials Interface

The Heat and Moisture Transport Interface

Heat and Moisture

The **Building Material** node is defined on the spatial frame. The material properties should be entered in the spatial frame, and the coupling with a moving frame interface is not supported. See Handling Frames in Heat Transfer for more details.

When **Surface-to-surface radiation** is activated, the Opacity subnode is automatically added to the entire selection, with **Opaque** option selected. The domain selection can't be edited. To set some part of the domain as transparent, add a new Opacity subnode from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Building Materials>Building Material

More locations are available if the **Heat transfer in porous media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Building Material

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Building Material**

Change Cross Section

Use this node with 1D components to model domains with another cross sectional area or another cross sectional perimeter than the global one that is used in the Heat Transfer interface **Physical Model** section. In 1D geometries, the temperature is assumed to be constant in the radial direction, and the heat equation is modified to account for that. See Equation 4-117 and Equation 4-118.

CHANGE CROSS SECTION

Enter values for the **Cross sectional area** A_c and the **Cross sectional perimeter** P_c to set the cross section of the domain in the plane perpendicular to the 1D geometry.

Out-of-Plane Heat Transfer

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Change Cross Section

Heat Transfer in Solids>Change Cross Section

Heat Transfer in Fluids>Change Cross Section
Heat Transfer in Porous Media>Change Cross Section
Heat Transfer in Building Materials>Change Cross Section
Bioheat Transfer>Change Cross Section
Heat Transfer with Surface-to-Surface Radiation>Change Cross Section

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Change Cross Section**

Change Thickness

Use this node with 2D components to model domains with another thickness than the overall thickness that is specified in the Heat Transfer interface **Physical Model** section. In 2D geometries, the temperature is assumed to be constant in the out-of-plane direction (z direction with default spatial coordinate names). The heat equation is modified to account for that. See Equation 4-113 and Equation 4-114.

CHANGE THICKNESS

Specify a value for the **Thickness** d_z of the domain in the out-of-plane direction. This value replaces the overall thickness in the domains that are selected in the **Domain Selection** section, and is used to multiply some terms into the heat equation.

Out-of-Plane Heat Transfer

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Change Thickness
Heat Transfer in Solids>Change Thickness
Heat Transfer in Fluids>Change Thickness
Heat Transfer in Porous Media>Change Thickness
Heat Transfer in Building Materials>Change Thickness
Bioheat Transfer>Change Thickness
Heat Transfer with Surface-to-Surface Radiation>Change Thickness
Heat Transfer with Radiation in Participating Media>Change Thickness

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Change Thickness**

Fluid

This node uses the following version of the heat equation to model heat transfer in fluids:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \qquad (6\text{-}3)$$
$$\mathbf{q} = -k \nabla T$$

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the fluid density.
- C_p (SI unit: J/(kg·K)) is the fluid heat capacity at constant pressure.
- k (SI unit: W/(m·K)) is the fluid thermal conductivity (a scalar or a tensor if the thermal conductivity is anisotropic).
- $\mathbf{u}$ (SI unit: m/s) is the fluid velocity field, either an analytic expression or a velocity field from a Fluid Flow interface.
- Q (SI unit: W/m^3) is the heat source (or sink). Add one or several heat sources as separate physics features. See Heat Source node, and Viscous Dissipation and Pressure Work subnodes for example.

For a steady-state problem the temperature does not change with time and the first term disappears.

MODEL INPUTS

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when temperature-dependent material properties are used. By default the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** ( ). The available options are **User defined** (default) and all temperature variables from the physics interfaces included in the model. These physics interfaces have their own tags (the **Name**). For example, if a **Heat Transfer in Fluids** interface is included in the model, the **Temperature (ht)** option is available.

Absolute Pressure

The absolute pressure is used in some predefined quantities that include the enthalpy (the energy flux, for example).

It is also used if the ideal gas law is applied. See Thermodynamics, Fluid.

The default **Absolute pressure** p_{A} is **User defined**. When additional physics interfaces are added to the model, the absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Laminar Flow** interface is added you can select **Absolute pressure (spf)** from the list.

Velocity Field

The default **Velocity field** $\mathbf{u}$ is **User defined**. For **User defined** enter values or expressions for the components based on space dimensions. Or select an existing velocity field in the component (for example, **Velocity field (spf)** from a **Laminar Flow** interface).

Concentration

From the **Concentration** c (SI unit: $\mathrm{mol/m^3}$ or $\mathrm{kg/m^3}$) list, select an existing concentration variable from another physics interface, if any concentration variables exist, or select **User defined** to enter a value or expression for the concentration. This section can be edited anytime a material property is concentration dependent; for example, when the **Fluid type** is set to **Moist air** with **Input quantity** set to **Concentration**.

FLUID MATERIAL

This section is available only when the **Local Thermal Non-Equilibrium** multiphysics coupling is included in the component to model porous media. It makes it possible to define different material properties for the fluid phase when the domain material corresponds to the solid phase (porous matrix) material.

Select any material from the list to define the **Fluid material**. The default uses the **Domain material**.

HEAT CONDUCTION, FLUID

The thermal conductivity k describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k\nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity, and enter values or expressions for the thermal conductivity or its components. For **Isotropic** enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

THERMODYNAMICS, FLUID

This section sets the thermodynamics properties of the fluid.

The heat capacity at constant pressure C_p describes the amount of heat energy required to produce a unit temperature change in a unit mass.

The ratio of specific heats γ is the ratio of the heat capacity at constant pressure, C_p, to the heat capacity at constant volume, C_v. When using the ideal gas law to describe a fluid, specifying γ is sufficient to evaluate C_p. For common diatomic gases such as air, $\gamma = 1.4$ is the standard value. Most liquids have $\gamma = 1.1$ while water has $\gamma = 1.0$. γ is used in the streamline stabilization and in the variables for heat fluxes and total energy fluxes. It is also used if the ideal gas law is applied.

The available **Fluid type** options are **Gas/Liquid** (default), **Moist air**, and **Ideal gas**. After selecting a **Fluid type** from the list, further settings display underneath.

Gas/Liquid

This option specifies the **Density**, the **Heat capacity at constant pressure**, and the **Ratio of specific heats** for a general gas or liquid.

Ideal Gas

This option uses the ideal gas law to describe the fluid. Only two properties are needed to define the thermodynamics of the fluid:

- The gas constant, with two options for the **Gas constant type**: **Specific gas constant** R_s or **Mean molar mass** M_n. If **Mean molar mass** is selected the software uses the universal gas constant $R = 8.314$ J/(mol·K), which is a built-in physical constant, to compute the specific gas constant.
- Either the **Heat capacity at constant pressure** C_p or **Ratio of specific heats** γ by selecting the option from the **Specify Cp or** γ list. For an ideal gas, it is sufficient to specify either C_p or the ratio of specific heats, γ, as these properties are interdependent.

Moist Air

If **Moist air** is selected, the thermodynamics properties are defined as a function of the quantity of vapor in the moist air. The available **Input quantity** options to define the amount of vapor in the moist air are the following:

- **Vapor mass fraction** to define the ratio of the vapor mass to the total mass. Enter a value or expression for the **Vapor mass fraction** ω.
- **Concentration** to define the amount of water vapor in the total volume. If selected, a **Concentration** model input is added in the **Model Inputs** section.
- **Moisture content** (the default), also called mixing ratio or humidity ratio, to define the ratio of the water vapor mass to the dry air mass. For **User defined**, enter a value or expression for the **Moisture Content** x_{vap}. Else, select an **Ambient moisture content** defined from the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.
- **Relative humidity**, by defining the **Reference relative humidity** at **Reference temperature** and **Reference pressure**. The **Reference relative humidity** is a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to a water vapor-saturated air. For **User defined**, enter a value or expression for the **Reference relative humidity** ϕ_{ref}. Else, select an **Ambient relative humidity** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface. For consistency, the **Reference temperature** and **Reference pressure** should also be taken from ambient conditions.

DYNAMIC VISCOSITY

This section is only available when the **Equivalent conductivity for convection** check box is selected in the **Equivalent conductivity for convection** section. The **Dynamic viscosity** μ is then used to compute the Nusselt number.

EQUIVALENT CONDUCTIVITY FOR CONVECTION

When the **Equivalent conductivity for convection** check box is selected, the fluid thermal conductivity is increased according to the Nusselt number to account for the contribution of the convective heat flux. In addition the user-defined or predefined velocity model input is ignored and the fluid velocity is set to zero. This check box is not selected by default and requires the Heat Transfer Module.

The options in the **Nusselt number correlation** list are:

- **Horizontal cavity heated from below**, for which values for the **Cavity height** H and the **Temperature difference** ΔT should be specified for the computation of the Nusselt number.
- **Vertical rectangular cavity**, for which values for the **Cavity height** H, the **Plate distance** L, and the **Temperature difference** ΔT should be specified for the computation of the Nusselt number.
- **User defined**, for which a value for Nu should be specified directly.

Moist Air Fluid Type

Local Thermal Non-Equilibrium

Theory for Heat Transfer in Fluids

Equivalent Thermal Conductivity Correlations

With certain COMSOL products, the Viscous Dissipation (for heat generated by viscous friction) and Pressure Work subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

When **Surface-to-surface radiation** is activated, the Opacity subnode is automatically added to the entire selection, with **Transparent** option selected. The domain selection can't be edited. To set some part of the domain as opaque, add a new Opacity subnode from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

Heat Sink: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/heat_sink**

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Fluid
Heat Transfer in Solids>Fluid
Heat Transfer in Fluids>Fluid
Heat Transfer in Porous Media>Fluid
Heat Transfer in Building Materials>Fluid
Bioheat Transfer>Fluid
Heat Transfer with Surface-to-Surface Radiation>Fluid
Heat Transfer with Radiation in Participating Media>Fluid

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Fluid**

Geothermal Heating

Use this node to account for the contribution of geothermal heat production by adding a source term Q_{geo} to the right-hand side of the heat equation:

$$Q_{\mathrm{geo}} = \rho_{\mathrm{geo}} q_{\mathrm{geo}} f(z_{\mathrm{geo}})$$

The predefined expression of the heat source uses the **Geothermal Density**, the **Radiogenic heating per unit mass**, and a distribution function that can be set.

GEOTHERMAL HEAT PRODUCTION

Specify the **Radiogenic heating per unit mass** q_{geo}.

The two option buttons in the same group control the distribution function $f(z_{\mathrm{geo}})$:

- If **Uniform distribution** is selected, $f(z_{\mathrm{geo}}) = 1$ and the geothermal heat source is assumed to be independent on depth.
- If **Exponential distribution** is selected, two additional input fields are displayed underneath. The exponential distribution $f(z_{\mathrm{geo}}) = \exp(z_{\mathrm{geo}}/h_{\mathrm{geo}})$, is defined by the constant **Length scale:** h_{geo} and the variable **Depth** z_{geo}. The depth can be, for example, the vertical coordinate direction. The heat source achieves its maximum value where $z_{\mathrm{geo}} = 0$, typically the top surface of a model.

GEOTHERMAL DENSITY

Select the **Geothermal density**: **Solids**, **Porous media**, or **User defined**:

- If **Solids** is selected, it calculates the geothermal density based on the volume fraction of solid material

$$\rho_{\mathrm{geo}} = \frac{\sum_i \theta_{\mathrm{p}i}\rho_{\mathrm{p}i}}{\sum_i \theta_{\mathrm{p}i}}$$

- If **Porous media** is selected, it calculates the geothermal density based on all mobile and immobile components of the porous medium:

$$\rho_{\mathrm{geo}} = \sum_i \theta_{\mathrm{p}i}\rho_{\mathrm{p}i} + \sum_i \theta_{\mathrm{g}i}\rho_{\mathrm{g}i} + \theta_{\mathrm{L}}\rho_{\mathrm{L}}$$

- If **User defined** is selected, enter a value for the **Geothermal density** ρ_{geo}.

The **Geothermal Heating** subnode requires the Subsurface Flow Module. For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

If the **Porous matrix model** is set to **Extended** under the **Physical Model** section:

Heat Transfer in Porous Media>Porous Medium>Geothermal Heating

More locations are available if the **Heat transfer in porous media** check box is selected and **Porous matrix model** is set to **Extended** under the **Physical Model** section. For example:

Heat Transfer in Solids>Porous Medium>Geothermal Heating

Ribbon

Physics Tab with **Porous Medium** selected in the model tree:

Attributes>Porous Medium>Geothermal Heating

Heat Source

This node describes heat generation within the domain. You express heating and cooling with positive and negative values, respectively. Add one or more nodes as needed — all heat sources within a domain contribute to the total heat source.

The **Heat Source** node adds a source term Q to the right-hand side of the heat equation:

$$Q = Q_0$$

Specify Q_0 as the heat per unit volume, as a linear heat source, or as a heat rate.

HEAT SOURCE

Click the **General source** (the default), **Linear source**, or **Heat rate** buttons.

- For **General source** enter a value for the distributed heat source Q_0 when the default option (**User defined**) is selected. See also Additional General Source Options to use predefined heat sources available from other interfaces.
- For **Linear source** enter a value for the **Production/absorption coefficient** q_s used in the predefined linear expression. The advantage of writing the source on this form is that it can be accounted for in the streamline diffusion stabilization. The stabilization applies when q_s is independent of the temperature, but some stability can be gained as long as q_s is only weakly dependent on the temperature.
- For **Heat rate** enter a value for the heat rate P_0. In this case $Q_0 = P_0/V$, where V is the total volume of the selected domains.

Additional General Source Options

For the general heat source Q_0 there are predefined heat sources available (in addition to a **User defined** heat source) when simulating heat transfer together with electrical or electromagnetic interfaces. Such sources represent, for example, ohmic heating and induction heating. Depending on additional physics interfaces, the following are available:

- With the addition of an Electric Currents interface, the **Total power dissipation density (ec)** heat source is available from the **General source** list.
- With the addition of any version of the Electromagnetic Waves interface (which requires the RF Module), the **Total power dissipation density (emw)** and **Electromagnetic power loss density (emw)** heat sources are available from the **General source** list.

- With the addition of a Magnetic Fields interface (a 3D component requires the AC/DC Module), the **Electromagnetic heating (mf)** heat source is available from the **General source** list.
- With the addition of a Magnetic and Electric Fields interface (which requires the AC/DC Module), the **Electromagnetic heating (mef)** heat source is available from the **General source** list.
- For the Heat Transfer in Porous Media interface, with the addition of interfaces from the Batteries & Fuel Cells Module, Corrosion Module, or Electrodeposition Module, heat sources from the electrochemical current distribution interfaces are available.

FRAME SELECTION

To display this section, add both a **Heat Transfer (ht)** and a **Moving Mesh (ale)** interface (found under the **Mathematics>Deformed Mesh** branch when adding a physics interface). Then click the **Show** button ( ) and select **Advanced Physics Options**.

When the model contains a moving mesh, the **Enable conversions between material and spatial frame** check box is selected by default in the Heat Transfer interface, which in turn enables further options. Use **Frame Selection** to select the frame where the input variables are defined. If **Spatial** is selected, the variables take their values from the text fields. If **Material** (the default) is selected, a conversion from the material to the spatial frame is applied to the text field values.

- Handling Frames in Heat Transfer
- Stabilization Techniques in the *COMSOL Multiphysics Reference Manual*

For the definition of a localized heat source, see Line Heat Source, Line Heat Source and Point Heat Source.

For the definition of a heat on a boundary, see Boundary Heat Source.

Forced Convection Cooling of an Enclosure with Fan and Grille: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/electronic_enclosure_cooling**

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Heat Source
Heat Transfer in Solids>Heat Source
Heat Transfer in Fluids>Heat Source
Heat Transfer in Porous Media>Heat Source
Heat Transfer in Building Materials>Heat Source
Bioheat Transfer>Heat Source
Heat Transfer with Surface-to-Surface Radiation>Heat Source
Heat Transfer with Radiation in Participating Media>Heat Source

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Heat Source**

Immobile Fluids

This node should be used to model an unsaturated porous matrix for which the pore space is not filled with liquid and contains one or more gas considered as immobile fluids; or a porous matrix containing insulated enclosures.

It is possible to add and define up to five different immobile fluids and volume fractions to the porous matrix.

IMMOBILE FLUIDS

In this section, the **Number of fluids** to define — **1**, **2**, **3**, **4**, or **5** — is set. Depending on the **Number of fluids**, further settings display underneath. For each fluid:

- The material must be set from the **Fluid material** list, which can point to any material in the model.
- The **Volume fraction** of the immobile fluid $\theta_{g\ \{1,2,3,4,5\}}$ should be set.

The total volume fraction of immobile fluids is calculated from

$$\theta_g = \sum_i \theta_{gi}$$

The volume fraction available for mobile fluids (that is, the effective porosity) is then calculated from

$$\theta_L = 1 - \theta_p - \theta_g$$

where the total volume fraction of immobile solids is calculated from

$$\theta_p = \sum_i \theta_{pi}$$

HEAT CONDUCTION

For the same number of fluids selected under **Immobile Fluids**, the defaults for the **Thermal conductivity** k_g use values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity and other values or expressions in the fields or matrices.

The effective conductivity for the equivalent immobile fluid is calculated from

$$k_g = \sum_i \theta_{gi} k_{gi}$$

When one or more than one solid is selected in the **Immobile Solids** section, the effective conductivity of immobile solids and immobile fluids can be calculated in three different ways:

- If **Volume average** is selected under **Effective Thermal Conductivity**, the effective conductivity of the solid-fluid system is given by

  $$k_{eff} = \theta_p k_p + \theta_g k_g + \theta_L k_L$$

  where k_p and k_g are the effective conductivities of immobile solids and fluids.
- If **Reciprocal average** is selected under **Effective Thermal Conductivity**, the effective conductivity is calculated from

$$\frac{1}{k_{\mathrm{eff}}} = \frac{\theta_{\mathrm{p}}}{k_{\mathrm{p}}} + \frac{\theta_{\mathrm{g}}}{k_{\mathrm{g}}} + \frac{\theta_{\mathrm{L}}}{k_{\mathrm{L}}}$$

where k_{p} and k_{g} are the effective conductivities of immobile solids and fluids.

- If **Power law** is selected under **Effective Thermal Conductivity**, the effective conductivity is calculated from

$$k_{\mathrm{eff}} = k_{\mathrm{p}}^{\theta_{\mathrm{p}}} \cdot k_{\mathrm{g}}^{\theta_{\mathrm{g}}} \cdot k_{\mathrm{L}}^{\theta_{\mathrm{L}}}$$

where k_{p} and k_{g} are the effective conductivities of immobile solids and fluids.

THERMODYNAMICS

For the same number of fluids selected under **Immobile Fluids**, the following properties should be set:

- **Density** $\rho_{\mathrm{g}\{1,2,3,4,5\}}$
- **Specific heat capacity** $C_{p,\mathrm{g}\{1,2,3,4,5\}}$

The effective volumetric heat capacity of the composite solid-fluid system is defined as

$$(\rho C_p)_{\mathrm{eff}} = \sum_i \theta_{\mathrm{p}i}\rho_{\mathrm{p}i}C_{p,\,\mathrm{p}i} + \sum_i \theta_{\mathrm{g}i}\rho_{\mathrm{g}i}C_{p,\,\mathrm{g}i} + \theta_{\mathrm{L}}\rho_{\mathrm{L}}C_{p,\,\mathrm{L}}$$

The **Immobile Fluids** node requires the Subsurface Flow Module. For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

If the **Porous matrix model** is set to **Extended** under the **Physical Model** section:

Heat Transfer in Porous Media>Porous Medium>Immobile Fluids

More locations are available if the **Heat transfer in porous media** check box is selected and **Porous matrix model** is set to **Extended** under the **Physical Model** section. For example:

Heat Transfer in Solids>Porous Medium>Immobile Fluids

Ribbon

Physics Tab with **Porous Medium** selected in the model tree:

Attributes>Porous Medium>Immobile Fluids

Initial Values

This node adds an initial value for the temperature that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. In addition to the default **Initial Values** node always present in the interface, you can add more **Initial Values** nodes if needed.

INITIAL VALUES

For **User defined**, enter a value or expression for the initial value of the **Temperature** T (SI unit: K). The default value is approximately room temperature, 293.15 K (20 °C). Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Initial Values
Heat Transfer in Solids>Initial Values
Heat Transfer in Fluids>Initial Values
Heat Transfer in Porous Media>Initial Values
Heat Transfer in Building Materials>Initial Values
Bioheat Transfer>Initial Values
Heat Transfer with Surface-to-Surface Radiation>Initial Values
Heat Transfer with Radiation in Participating Media>Initial Values

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface*>Initial Values

Isothermal Domain

This node should be used when the temperature shows sufficiently small spatial fluctuations to be considered homogeneous in space but not necessarily in time.

ISOTHERMAL DOMAIN

Two options are available for the **Temperature definition** into the **Isothermal Domain**:

- If **From heat balance** (the default) is selected, the temperature is computed as the solution of a reduced form of the heat equation due to spatial homogeneity. See Equation 4-29.
- If **From prescribed temperature** is selected, the temperature is set to the domain **Temperature** T_0 that needs to be specified.

THERMODYNAMICS

The thermodynamics properties of the **Isothermal Domain** are set in this section. Two options are available for the **Mass definition**:

- If **Density** is selected, the **Density** ρ should be specified.
- If **Total mass** is selected, the **Mass** m should be specified.

Finally the **Heat capacity at constant pressure** C_p should be specified.

It is not possible to couple an **Isothermal Domain** with the **Laminar Flow** interface through the **Non-Isothermal Flow** multiphysics condition.

The following conditions of heat exchange can be applied at isothermal domain interfaces: thermal insulation, continuity, ventilation, convective heat flux, and thermal contact. See Isothermal Domain Interface for more details.

Also see Theory for Lumped Isothermal Domain.

When **Surface-to-surface radiation** is activated, the Opacity subnode is automatically added to the entire selection, with **Transparent** option selected. The domain selection can’t be edited. To set some part of the domain as opaque, add a new Opacity subnode from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

	Natural Convection Cooling of a Vacuum Flask: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/vacuum_flask**

LOCATION IN USER INTERFACE

Context menus

If the **Isothermal domain** check box is selected under the **Physical Model** section:

Heat Transfer>Isothermal Domain

Heat Transfer in Solids>Isothermal Domain

Heat Transfer in Fluids>Isothermal Domain

Heat Transfer in Porous Media>Isothermal Domain

Heat Transfer in Building Materials>Isothermal Domain

Bioheat Transfer>Isothermal Domain

Heat Transfer with Surface-to-Surface Radiation>Isothermal Domain

Heat Transfer with Radiation in Participating Media>Isothermal Domain

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Isothermal Domain**

Opacity

This subnode enables to define the surface-to-surface radiation direction on boundaries surrounding the domains where it is defined. When several spectral bands are defined, the opacity can be set for all or only a selection of wavelengths.

When the **Radiation direction** is defined by **Opacity controlled** in surface-to-surface boundary features (Diffuse Mirror, Diffuse Surface, Prescribed Radiosity), surface-to-surface radiation propagates in non-opaque domains. Alternatively the **Radiation direction** can be defined using the normal orientation or on both sides of boundaries. In this case the **Opacity** node is ignored.

OPACITY

Select **Opaque** or **Transparent** to set the domain's opacity type. Depending on the **Wavelength dependence of emissivity** defined in the Radiation Settings section of the physics interface settings, different sections display underneath:

- When **Wavelength dependence of emissivity** is set to **Constant** in the physics interface settings, set the opacity for all wavelengths.
- When **Wavelength dependence of emissivity** is set to **Solar and ambient** or **Multiple spectral bands** in the physics interface settings, set the opacity for each spectral band by selecting the corresponding check box in **Opacity on solar spectral band** and **Opacity on ambient spectral band**, or **Opacity on spectral band i** sections. By default the opacity is set to the same type for all spectral bands.

The default opacity type depends on the parent node's type. It is set to **Opaque** for Solid, Porous Medium, Biological Tissue and Building Material domain features and Surface-to-Surface Radiation interface; and to **Transparent** for Fluid, Phase Change Material, and Isothermal Domain domain features.

- The **Wavelength dependence of emissivity** is defined in the physics interface settings, in the Radiation Settings section.
- If this feature is combined with heat transfer in 2D and 1D, the thickness is assumed to be infinite for the view factor computation. The user-defined value for d is still used in the heat transfer equation.

Free Convection in a Light Bulb: Application Library path **Heat_Transfer_Module/Thermal_Radiation/light_bulb**

Thermo-Photo-Voltaic Cell: Application Library path **Heat_Transfer_Module/Thermal_Radiation/tpv_cell**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Surface-to-Surface Radiation>Solid>Opacity
Heat Transfer with Surface-to-Surface Radiation>Fluid>Opacity
Heat Transfer with Surface-to-Surface Radiation>Porous Medium>Opacity
Heat Transfer with Surface-to-Surface Radiation>Phase Change Material>Opacity
Heat Transfer with Surface-to-Surface Radiation>Biological Tissue>Opacity
Heat Transfer with Surface-to-Surface Radiation>Building Material>Opacity

Heat Transfer with Surface-to-Surface Radiation>Isothermal Domain>Opacity
Surface-to-Surface Radiation>Opacity

More locations are available if the **Surface-to-surface radiation** check box is selected under the **Physical Model** section. For example:
Heat Transfer in Solids>Solid>Opacity

Ribbon

Physics Tab with **Solid**, **Fluid**, **Porous Medium**, **Phase Change Material**, **Biological Tissue**, **Building Material**, or **Isothermal Domain** selected in the model tree:

Attributes>Opacity

Physics Tab with **Surface-to-Surface Radiation** selected in the model tree:

Domains>Opacity

Out-of-Plane Heat Flux

Out-of-plane heat transfer mechanism is used to reduce a model geometry to 2D or even 1D when the temperature variation is small in one or more directions; for example, when the object to model is thin or slender. For the obtained 1D and 2D components, this node adds a heat flux $q_{0,\mathrm{u}}$ for the upside heat flux and a heat flux $q_{0,\mathrm{d}}$ for the downward heat flux to the right-hand side of the heat equation.

For example, in 2D components, heat transfer in solids and heat transfer in fluids are given by Equation 6-4 and Equation 6-5:

$$d_z\rho C_p\frac{\partial T}{\partial t} - \nabla\cdot\mathbf{q} = d_zQ + q_0 \tag{6-4}$$

$$d_z\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u}\cdot\nabla T\right) = \nabla\cdot\mathbf{q} + d_zQ + q_0 \tag{6-5}$$

$$q_0 = q_{0,\mathrm{u}} + q_{0,\mathrm{d}}$$

The convective heat flux adds the following contribution

$$q_0 = h_\mathrm{u}(T_{\mathrm{ext,u}} - T) + h_\mathrm{d}(T_{\mathrm{ext,d}} - T)$$

UPSIDE INWARD HEAT FLUX

The available options are **General inward heat flux** and **Convective heat flux**. The settings are the same as for the Heat Flux node.

DOWNSIDE INWARD HEAT FLUX

The available options are **General inward heat flux** and **Convective heat flux**. The settings are the same as for the Heat Flux node.

See Out-of-Plane Heat Transfer for the formulation of out-of-plane heat transfer in 1D, 1D axisymmetric, and 2D geometries.

See also Out-of-Plane Domain Fluxes.

Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.

See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

Out-of-Plane Heat Transfer for a Thin Plate: Application Library path **Heat_Transfer_Module/Verification_Examples/thin_plate**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Out-of-Plane Heat Flux
Heat Transfer in Solids>Out-of-Plane Heat Flux
Heat Transfer in Fluids>Out-of-Plane Heat Flux
Heat Transfer in Porous Media>Out-of-Plane Heat Flux
Heat Transfer in Building Materials>Out-of-Plane Heat Flux
Bioheat Transfer>Out-of-Plane Heat Flux
Heat Transfer with Radiation in Participating Media>Out-of-Plane Heat Flux

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Out-of-Plane Heat Flux**

Out-of-Plane Radiation

Out-of-plane heat transfer mechanism is used to reduce a model geometry to 2D or even 1D when the temperature variation is small in one or more directions; for example, when the object to model is thin or slender. This node models surface-to-ambient radiation on the upside and downside for the obtained 1D and 2D components. It adds the following contribution to the right-hand side of Equation 6-4 or Equation 6-5:

$$\varepsilon_{\mathrm{u}}\sigma(T_{\mathrm{amb,\,u}}^4 - T^4) + \varepsilon_{\mathrm{d}}\sigma(T_{\mathrm{amb,\,d}}^4 - T^4)$$

UPSIDE PARAMETERS

Surface emissivity

The default **Surface emissivity** ε_{u} (a dimensionless number between 0 and 1) is taken **From material**. For **User defined**, it should be specified. An emissivity of 0 means that the surface emits no radiation at all while an emissivity of 1 means that it is a perfect blackbody. The default is 0.

Ambient temperature

For **User defined**, enter a value or expression for the **Ambient temperature** $T_{\mathrm{amb,\,u}}$. The default value is approximately room temperature, 293.15 K (20 °C). Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

DOWNSIDE PARAMETERS

Follow the instructions for the **Upside Parameters** section to define the downside parameters ε_{d} and $T_{\mathrm{amb,\,d}}$.

Out-of-Plane Heat Transfer

Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.

See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

Out-of-Plane Heat Transfer for a Thin Plate: Application Library path **Heat_Transfer_Module/Verification_Examples/thin_plate**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Out-of-Plane Radiation
Heat Transfer in Solids>Out-of-Plane Radiation
Heat Transfer in Fluids>Out-of-Plane Radiation
Heat Transfer in Porous Media>Out-of-Plane Radiation
Heat Transfer in Building Materials>Out-of-Plane Radiation
Bioheat Transfer>Out-of-Plane Radiation
Heat Transfer with Radiation in Participating Media>Out-of-Plane Radiation

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer** or **Heat Transfer with Radiation in Participating Media** selected:
Domains>*interface***>Out-of-Plane Radiation**

Phase Change Material

This node should be used to solve the heat equation after specifying the properties of a phase change material according to the apparent heat capacity formulation. This formulation gets its name from the fact that the latent heat is included as an additional term in the heat capacity. Up to five transitions in phase per material are supported.

NUMBER OF TRANSITIONS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. The **Number of phase transitions** to model is set in this section. In most cases, only one phase transition is needed to simulate solidification, melting, or evaporation. If you want to model successive melting and evaporation, or any couple of successive phase transformations, choose 2 in the **Number of phase transitions** list. The maximum value is 5.

Depending on the **Number of phase transitions**, several parts display in the **Phase Change** section, and several **Phase** sections display underneath.

PHASE CHANGE

The parameters for the definition of the transition temperature intervals are set in this section.

Each transition is assumed to occur smoothly in a temperature interval between $T_{\mathrm{pc}, j \to j+1} - \Delta T_{j \to j+1}/2$ and $T_{\mathrm{pc}, j \to j+1} + \Delta T_{j \to j+1}/2$, releasing a total heat per unit volume equal to $L_{j \to j+1}$.

The **Phase change temperature between phase 1 and phase 2** $T_{\mathrm{pc}, 1 \to 2}$ should be set to define the center of the first transition interval. The default is 273.15 K. Enter any additional phase change temperatures as per the **Number of phase transitions**.

The **Transition interval between phase 1 and phase 2** $\Delta T_{1 \to 2}$ should be set to define the width of the first transition interval. The default is 10 K. Enter any additional transition intervals as per the **Number of phase transitions**.

The value of $\Delta T_{j \to j+1}$ must be strictly positive. A value near 0 K corresponds to a behavior close to a pure substance.

The **Latent heat from phase 1 and phase 2** $L_{1 \to 2}$ should be set to define the total heat per unit volume released during the first transition. Enter any additional latent heat values as per the **Number of phase transitions**.

The value of $L_{j \to j+1}$ must be positive. The default is 333 kJ/kg, which corresponds to the latent heat of fusion of water at a pressure of 1 atm.

About the Phases

The different phases are ordered according to the temperatures of fusion. Hence, the material properties of phase 1 are valid when $T < T_{\mathrm{pc}, 1 \to 2}$ while the material properties of phase 2 hold for $T > T_{\mathrm{pc}, 1 \to 2}$.

When more than one transition is modeled, the number of phases exceeds 2 and new variables are created (for example, $T_{\mathrm{pc}, 2 \to 3}$, $\Delta T_{2 \to 3}$ or $L_{2 \to 3}$). The phase change temperatures $T_{\mathrm{pc}, j \to j+1}$ are increasing and satisfy

$$T_{\mathrm{pc}, 1 \to 2} < T_{\mathrm{pc}, 2 \to 3} < \ldots$$

This defines distinct domains of temperature bounded by $T_{\mathrm{pc}, j-1 \to j}$ and $T_{\mathrm{pc}, j \to j+1}$ where the material properties of phase j only apply.

In addition, the values of $\Delta T_{j \to j+1}$ are chosen so that the ranges between $T_{\mathrm{pc}, j \to j+1} - \Delta T_{j \to j+1}/2$ and $T_{\mathrm{pc}, j \to j+1} + \Delta T_{j \to j+1}/2$ do not overlap. If this condition is not satisfied, unexpected behavior can occur because some phases would never form completely. The values of $\Delta T_{j \to j+1}$ must all be strictly positive.

PHASE

In each **Phase** section (based on the **Number of phase transitions**), the thermal conductivity and thermodynamics properties of each phase must be set. Then, within the transition interval, there is a "mushy zone" with mixed material properties.

Select a **Material, phase** [1,2,...], which can point to any material in the model. The default uses the **Domain material**.

The following material properties should be set:

- **Thermal conductivity** $k_{\mathrm{phase}[1,2,...]}$. The default uses the material values for phase i. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity, and enter another value or expression. The default is 1 W/(m·K).
- **Density** $\rho_{\mathrm{phase}[1,2,...]}$. The default is 1000 kg/m^3.
- **Heat capacity at constant pressure** $C_{p,\,\mathrm{phase}[1,2,...]}$. The default is 4200 J/(kg·K).
- **Ratio of specific heats** $\gamma_{\mathrm{phase}[1,2,...]}$. The default is 1.1

Theory for Heat Transfer with Phase Change

It is useful to choose three or more phase transitions to handle extra changes of material properties such as in mixtures of compounds, metal alloys, composite materials, or allotropic varieties of a substance. For example, α, γ, and δ-iron are allotropes of solid iron that can be considered as phases with distinct phase change temperatures.

When **Surface-to-surface radiation** is activated, the Opacity subnode is automatically added to the entire selection, with **Transparent** option selected. The domain selection can't be edited. To set some part of the domain as opaque, add a new Opacity subnode from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

To satisfy energy and mass conservation in phase change models, particular attention should be paid to the density in time simulations. When the material density is not constant over time — for example, dependent on the temperature — volume change is expected. The transport velocity field and the density must be defined so that mass is conserved locally. A Moving Mesh Interface (described in the *COMSOL Multiphysics Reference Manual*) can be used to account for model deformation.

Phase Change: Application Library path **Heat_Transfer_Module/Phase_Change/phase_change**

Continuous Casting: Application Library path **Heat_Transfer_Module/Thermal_Processing/continuous_casting**

Cooling and Solidification of Metal: Application Library path **Heat_Transfer_Module/Thermal_Processing/cooling_solidification_metal**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Phase Change Material
Heat Transfer in Solids>Phase Change Material
Heat Transfer in Fluids>Phase Change Material
Heat Transfer in Porous Media>Phase Change Material
Heat Transfer in Building Materials>Phase Change Material
Bioheat Transfer>Phase Change Material
Heat Transfer with Surface-to-Surface Radiation>Phase Change Material
Heat Transfer with Radiation in Participating Media>Phase Change Material

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**,

Heat Transfer with Surface-to-Surface Radiation or **Heat Transfer with Radiation in Participating Media** selected:
Domains>*interface***>Phase Change Material**

Porous Medium

This node uses the following version of the heat equation to model heat transfer in a porous matrix filled with a fluid:

$$(\rho C_p)_{\text{eff}}\frac{\partial T}{\partial t} + \rho C_p\mathbf{u}\cdot\nabla T + \nabla\cdot\mathbf{q} = Q \tag{6-6}$$

$$\mathbf{q} = -k_{\text{eff}}\nabla T \tag{6-7}$$

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the fluid density.
- C_p (SI unit: J/(kg·K)) is the fluid heat capacity at constant pressure.
- $(\rho C_p)_{\text{eff}}$ (SI unit: $J/(m^3{\cdot}K)$) is the effective volumetric heat capacity at constant pressure defined by an averaging model to account for both solid matrix and fluid properties.
- $\mathbf{q}$ is the conductive heat flux (SI unit: W/m^2).
- $\mathbf{u}$ (SI unit: m/s) is the fluid velocity field, either an analytic expression or the velocity field from a Fluid Flow interface. $\mathbf{u}$ should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area. The average linear velocity (the velocity within the pores) can be calculated as $\mathbf{u}_{\text{L}} = \mathbf{u}/\theta_{\text{L}}$, where θ_{L} is the fluid's volume fraction, or equivalently the porosity.
- k_{eff} (SI unit: W/(m·K)) is the effective thermal conductivity (a scalar or a tensor if the thermal conductivity is anisotropic), defined by an averaging model to account for both solid matrix and fluid properties.
- Q (SI unit: W/m^3) is the heat source (or sink). Add one or several heat sources as separate physics features. See Heat Source node and Viscous Dissipation subnode for example.

For a steady-state problem the temperature does not change with time and the first term disappears.

MODEL INPUTS

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when temperature-dependent material properties are used. By default the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** ( ). The available options are **User defined** (default) and all the temperature variables from the physics interfaces included in the model. These physics interfaces have their own tags (the **Name**). For example, if a **Heat Transfer in Fluids** interface is included in the model, the **Temperature (ht)** option is available.

Absolute Pressure

The absolute pressure is used in some predefined quantities that include the enthalpy (the energy flux, for example).

It is also used if the ideal gas law is applied. See Thermodynamics, Fluid.

The default **Absolute pressure** p_A is **User defined**. When additional physics interfaces are added to the model, the absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Brinkman Equations** interface is added you can select **Absolute pressure (br)** from the list.

Velocity Field

The default **Velocity field** $\mathbf{u}$ is **User defined**. For **User defined** enter values or expressions for the components based on space dimensions. Or select an existing velocity field in the component (for example, **Velocity field (br)** from a **Brinkman Equations** interface).

Concentration

From the **Concentration** c (SI unit: mol/m^3 or kg/m^3) list, select an existing concentration variable from another physics interface, if any concentration variables exist, or select **User defined** to enter a value or expression for the concentration. This section can be edited anytime a material property is concentration dependent; for example, when the **Fluid type** is set to **Moist air** with **Input quantity** set to **Concentration**.

FLUID MATERIAL

Select any component material from the list to define the **Fluid material**. The default uses the **Domain material**. It makes it possible to define different material properties for the fluid phase when the domain material corresponds to the solid phase (porous matrix) material.

HEAT CONDUCTION, FLUID

The thermal conductivity k describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k\nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic** enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

THERMODYNAMICS, FLUID

This section sets the thermodynamics properties of the fluid.

The heat capacity at constant pressure C_p describes the amount of heat energy required to produce a unit temperature change in a unit mass.

The ratio of specific heats γ is the ratio of the heat capacity at constant pressure, C_p, to the heat capacity at constant volume, C_v. When using the ideal gas law to describe a fluid, specifying γ is sufficient to evaluate C_p. For common diatomic gases such as air, $\gamma = 1.4$ is the standard value. Most liquids have $\gamma = 1.1$ while water has $\gamma = 1.0$. γ is used in the streamline stabilization and in the variables for heat fluxes and total energy fluxes. It is also used if the ideal gas law is applied.

The available **Fluid type** options are **Gas/Liquid** (default), **Moist air**, or **Ideal gas**. After selecting a **Fluid type** from the list, further settings display underneath.

Gas/Liquid

This option specifies the **Density**, the **Heat capacity at constant pressure**, and the **Ratio of specific heats** for a general gas or liquid.

Ideal Gas

This option uses the ideal gas law to describe the fluid. Only two properties are needed to define the thermodynamics of the fluid:

- The gas constant, with two options for the **Gas constant type**: **Specific gas constant** R_s or **Mean molar mass** M_n. If **Mean molar mass** is selected the software uses the universal gas constant R = 8.314 J/(mol·K), which is a built-in physical constant, to compute the specific gas constant.

- Either the **Heat capacity at constant pressure** C_p or **Ratio of specific heats** γ by selecting the option from the **Specify Cp or** γ list. For an ideal gas, it is sufficient to specify either C_p or the ratio of specific heats, γ, as these properties are dependent.

Moist Air

If **Moist air** is selected, the thermodynamics properties are defined as a function of the quantity of vapor in the moist air. The available **Input quantity** options to define the amount of vapor in the moist air are the following:

- **Vapor mass fraction** (the default) to define the ratio of the vapor mass to the total mass.
- **Concentration** to define the amount of water vapor in the total volume. If selected, a **Concentration** model input is automatically added in the **Model Inputs** section.
- **Moisture content** (also called mixing ratio or humidity ratio) to define the ratio of the water vapor mass to the dry air mass.
- **Relative humidity**, by defining the **Reference relative humidity** at **Reference temperature** and **Reference pressure**. The **Reference relative humidity** is a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to a water vapor-saturated air.

IMMOBILE SOLIDS

This section sets the material and volume fraction of the porous matrix.

If the **Standard** porous matrix model is selected under **Physical Model**, select any component material in the **Solid material** list. The **Volume fraction** θ_p for the solid material should be specified.

If the **Extended** porous matrix model is selected under **Physical Model** (with the Subsurface Flow Module), the **Number of solids** can be set from **1** to **5**. Then for each solid a **Solid material** list and a **Volume fraction** field display underneath.

The total volume fraction of solid material is given by

$$\theta_\mathrm{p} = \sum_i \theta_{\mathrm{p}i}$$

and the available volume fraction for the mobile fluid is defined as

$$\theta_{\mathrm{L}} = 1-\sum_i \theta_{\mathrm{p}i}$$

HEAT CONDUCTION, POROUS MATRIX

The thermal conductivity k_{p} describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k_{\mathrm{p}}\nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k_{p} is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic** enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

When the **Extended** porous matrix model is selected under **Physical Model** (with the Subsurface Flow Module), and more than one solid is selected in the **Immobile Solids** section, the thermal conductivities $k_{\mathrm{p}i}$ should be specified for each immobile solid. The average property for the porous matrix is given by:

$$k_{\mathrm{p}} = \sum_i \theta_{\mathrm{p}i} k_{\mathrm{p}i}$$

THERMODYNAMICS, POROUS MATRIX

This section sets the thermodynamics properties of the porous matrix.

The specific heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass of the solid material.

The **Density** ρ_{p} and the **Specific heat capacity** $C_{p,\,\mathrm{p}}$ should be specified.

The effective volumetric heat capacity of the solid-liquid system is calculated from

$$(\rho C_p)_{\mathrm{eff}} = \theta_{\mathrm{p}}\rho_{\mathrm{p}}C_{p,\,\mathrm{p}} + (1-\theta_{\mathrm{p}})\rho C_p$$

When the **Extended** porous matrix model is selected under **Physical Model** (with the Subsurface Flow Module), and more than one solid is selected in the **Immobile Solids** section, the **Density** and **Specific heat capacity** should be specified for each immobile solid.

The effective volumetric heat capacity of the composite solid-fluid system is defined as

$$(\rho C_p)_{\mathrm{eff}} = \sum_i \theta_{\mathrm{p}i}\rho_{\mathrm{p}i}C_{p,\,\mathrm{p}i} + \left(1-\sum_i \theta_{\mathrm{p}i}\right)\rho C_p$$

EFFECTIVE THERMAL CONDUCTIVITY

This section sets the averaging model for the computation of the **Effective conductivity** by accounting for both solid matrix and fluid properties. The following options are available with either the Subsurface Flow Module or the Heat Transfer Module:

- **Volume average** (default), which computes the effective conductivity of the solid-fluid system as the weighted arithmetic mean of fluid and porous matrix conductivities:

$$k_{\mathrm{eff}} = \theta_{\mathrm{p}}k_{\mathrm{p}} + (1-\theta_{\mathrm{p}})k$$

- **Reciprocal average**, which computes the effective conductivity of the solid-fluid system as the weighted harmonic mean of fluid and porous matrix conductivities:

$$\frac{1}{k_{\mathrm{eff}}} = \frac{\theta_{\mathrm{p}}}{k_{\mathrm{p}}} + \frac{1-\theta_{\mathrm{p}}}{k}$$

- **Power law**, which computes the effective conductivity of the solid-fluid system as the weighted geometric mean of fluid and porous matrix conductivities:

$$k_{\mathrm{eff}} = k_{\mathrm{p}}^{\theta_{\mathrm{p}}} \cdot k^{(1-\theta_{\mathrm{p}})}$$

When the **Extended** porous matrix model is selected under **Physical Model** (with the Subsurface Flow Module), and more than one solid is selected in the **Immobile Solids** section, these averaging models are modified in the following way:

- Volume average:

$$k_{\mathrm{eff}} = \sum_i \theta_{\mathrm{p}i}k_{\mathrm{p}i} + \left(1-\sum_i \theta_{\mathrm{p}i}\right)k$$

- Reciprocal average:

$$\frac{1}{k_{\mathrm{eff}}} = \sum_i \theta_{\mathrm{p}i} \frac{\sum_i \theta_{\mathrm{p}i}}{\sum_i \theta_{\mathrm{p}i} k_{\mathrm{p}i}} + \left(1 - \sum_i \theta_{\mathrm{p}i}\right) \frac{1}{k}$$

- Power law:

$$k_{\mathrm{eff}} = \left(\frac{\sum_i \theta_{\mathrm{p}i} k_{\mathrm{p}i}}{\sum_i \theta_{\mathrm{p}i}} \right)^{\sum_i \theta_{\mathrm{p}i}} \cdot k^{\left(1 - \sum_i \theta_{\mathrm{p}i}\right)}$$

	Moist Air Fluid Type Local Thermal Non-Equilibrium Theory for Heat Transfer in Porous Media

	With certain COMSOL products, the Thermal Dispersion, Viscous Dissipation, Geothermal Heating and Immobile Fluids subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

	When **Surface-to-surface radiation** is activated, the Opacity subnode is automatically added to the entire selection, with **Opaque** option selected. The domain selection can't be edited. To set some part of the domain as transparent, add a new Opacity subnode from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

	Evaporation in Porous Media with Small Evaporation Rates: Application Library path **Heat_Transfer_Module/Phase_Change/evaporation_porous_media_small_rate**

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Porous Media>Porous Medium

More locations are available if the **Heat transfer in porous media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Porous Medium

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Porous Medium**

Pressure Work

This subnode adds the following contribution to the right-hand side of the **Heat Transfer in Fluids** equation to model the result of heating under adiabatic compression as well as some thermoacoustic effects:

$$Q_p = \alpha_p T\left(\frac{\partial p_{\mathrm{A}}}{\partial t} + \mathbf{u} \cdot \nabla p_{\mathrm{A}}\right) \tag{6-8}$$

where α_p is the coefficient of thermal expansion defined as:

$$\alpha_p = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p$$

The pressure work is computed using the relative pressure, and is generally small for low Mach number flows.

No settings are required.

When the **Non-Isothermal Flow** multiphysics coupling node is added, the effect of pressure work can be taken into account by selecting the **Include work done by pressure changes** check box under the **Flow Heating** section. In this case, the **Pressure Work** feature is overridden by the multiphysics coupling node's contribution.

Theory for Heat Transfer in Fluids

A similar term can be included to account for thermoelastic effects in solids. See Thermoelastic Damping.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Fluids>Fluid>Pressure Work

More locations are available. For example:

Heat Transfer in Solids>Fluid>Pressure Work

Ribbon

Physics Tab with **Fluid** selected in the model tree:

Attributes>Pressure Work

Radiation in Participating Media (Heat Transfer Interface)

This node should be used when radiation occurs in a medium not completely transparent, in which the radiation rays interact with the medium. It computes and adds in the heat transfer equation the radiative heat source term Q_r (SI unit: W/m^3), defined by:

$$Q_r = \kappa(G - 4\pi I_b(T))$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2), defined by

$$G = \int_{4\pi} I(\Omega) d\Omega$$

- $I(\Omega)$ is the radiative intensity (SI unit: W/(m^2·sr)) at a given position following the Ω direction, that satisfies the *radiative transfer equation*

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega'$$

- $I_b(T)$ is the blackbody radiative intensity (SI unit: W/(m^2·sr)), defined as

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi}$$

- n_r is the refractive index (dimensionless).
- σ is the Stefan-Boltzmann constant (SI unit: W/(m^2·K^4)).
- $\beta = \kappa + \sigma_s$ is the extinction coefficient (SI unit: 1/m).
- σ_s is the scattering coefficient (SI unit: 1/m).
- $\phi(\Omega', \Omega)$ is the scattering phase function (dimensionless).
- T is the temperature (SI unit: K).

It takes into account the absorbed and the emitted radiation, and depending on approximation method, also the scattered radiation.

Three approximation methods are available for the radiation discretization method. The characteristics of each of them are summarized in the following table.

TABLE 6-1: DISCRETIZATION METHODS FOR RADIATION IN PARTICIPATING MEDIA (HT INTERFACE)

OPTION	DISCRETE ORDINATES METHOD	P1 APPROXIMATION	ROSSELAND APPROXIMATION
Optical thickness validity	All	τ>>1	τ>>1
Scattering modeling	Linear Polynomial	Linear	No

TABLE 6-1: DISCRETIZATION METHODS FOR RADIATION IN PARTICIPATING MEDIA (HT INTERFACE)

OPTION	DISCRETE ORDINATES METHOD	P1 APPROXIMATION	ROSSELAND APPROXIMATION
Computation of G	$G \approx \sum_{i=1}^{N} \omega_i I_i$	$\nabla \cdot (D_{\mathrm{P1}} \nabla G) + \kappa(G - 4\pi I_{\mathrm{b}}) = 0$	G not computed. Thermal conductivity modified with $k_{\mathrm{R}} = \frac{16 n_{\mathrm{r}}^2 \sigma T^3}{3\beta_{\mathrm{R}}}$
Computational cost	High: up to 80 additional degrees of freedom (I_i)	Medium: 1 additional degree of freedom (G)	Low: No additional degree of freedom

MODELS INPUTS

There is one standard model input — the **Temperature** T. The default is to use the heat transfer interface's dependent variable.

RADIATION IN PARTICIPATING MEDIA

This section sets the absorption and scattering properties of the participating medium.

It is available when **Rosseland approximation** is selected as the **Radiation discretization method** for the physics interface. Depending on the available quantities, the extinction coefficient β_{R} can be specified directly or defined as the sum of the absorption and scattering coefficients. Also see Rosseland Approximation Theory.

The following options are available from the **Specify media properties** list:

- **Absorption and scattering coefficients** (default): in this case β_{R} is defined as $\beta_{\mathrm{R}} = \kappa + \sigma_{\mathrm{s}}$ and the **Absorption** and **Scattering** sections display underneath.
- **Extinction coefficient**: the default **Rosseland mean extinction coefficient** β_{R} should be specified directly.

ABSORPTION

This section sets the absorption property of the participating medium, and is available in the following cases:

- **Discrete ordinates method** is selected as the **Radiation discretization method**, or
- **P1 approximation** is selected as the **Radiation discretization method**, or
- **Rosseland approximation** is selected as the **Radiation discretization method**, and **Absorption and scattering coefficients** is selected from the **Specify media properties** list.

The **Absorption coefficient** κ should be specified. It defines the amount of radiation, $\kappa I(\Omega)$, that is absorbed by the medium.

SCATTERING

This section sets the scattering property of the participating medium, and is available in the following cases:

- **Discrete ordinates method** is selected as the **Radiation discretization method**, or
- **P1 approximation** is selected as the **Radiation discretization method**, or
- **Rosseland approximation** is selected as the **Radiation discretization method**, and **Absorption and scattering coefficients** is selected from the **Specify media properties** list.

The **Scattering coefficient** σ_s should be specified.

When **Discrete ordinates method** or **P1 approximation** is selected as the **Radiation discretization method** for the physics interface, choose in addition the **Scattering type**: **Isotropic**, **Linear anisotropic**, or **Polynomial anisotropic** (only with **Discrete ordinates method**). This provides options to approximate the scattering phase function using the cosine of the scattering angle, μ_0:

- **Isotropic** (the default) corresponds to the scattering phase function $\phi(\mu_0) = 1$.
- For **Linear anisotropic** it defines the scattering phase function as $\phi(\mu_0) = 1 + a_1\mu_0$. Enter the **Legendre coefficient** a_1.
- For **Polynomial anisotropic** it defines the scattering phase function as

$$\phi(\mu_0) = 1 + \sum_{m=1}^{12} a_m P_m(\mu_0)$$

  Enter each **Legendre coefficient** $a_1, \ldots, a_{12}$ as required.

INITIAL VALUES

When **Discrete ordinates method** is selected as the **Radiation discretization method** for the physics interface, the **Initial radiative intensity** I should be specified. The default is `ht.Ibinit`, which is the blackbody radiative intensity at initial temperature.

When **P1 approximation** is selected as the **Radiation discretization method** for the physics interface, the **Initial incident radiation** G should be specified. The default is `(4*pi)*ht.Ibinit`, computed from the blackbody radiative intensity at initial temperature.

This section is not available when **Rosseland approximation** is selected as the **Radiation discretization method** for the physics interface.

Theory for Radiation in Participating Media

Discrete Ordinates Method (DOM)

Rosseland Approximation Theory

P1 Approximation Theory

Radiative Cooling of a Glass Plate: Application Library path **Heat_Transfer_Module/Thermal_Radiation/glass_plate**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Radiation in Participating Media>Radiation in Participating Media

More locations are available if the **Radiation in participating media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Radiation in Participating Media

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Radiation in Participating Media**

Radiation in Participating Media (RPM Interface)

This node should be used when radiation occurs in a medium not completely transparent, in which the radiation rays interact with the medium. It computes the radiative heat source term Q_r (SI unit: $\mathrm{W/m^3}$), defined by:

$$Q_\mathrm{r} = \kappa(G - 4\pi I_\mathrm{b}(T))$$

where

- κ is the absorption coefficient (SI unit: $\mathrm{m^{-1}}$).

- G is the incident radiation (SI unit: W/m^2), defined by

$$G = \int_{4\pi} I(\Omega)d\Omega$$

- $I(\Omega)$ is the radiative intensity (SI unit: $W/(m^2 \cdot sr)$) at a given position following the Ω direction, that satisfies the *radiative transfer equation*

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi}\int_{4\pi} I(\Omega')\phi(\Omega', \Omega)d\Omega'$$

- $I_b(T)$ is the blackbody radiative intensity (SI unit: $W/(m^2 \cdot sr)$), defined as

$$I_b(T) = \frac{n_r^{\ 2}\sigma T^4}{\pi}$$

- n_r is the refractive index (dimensionless).
- σ is the Stefan-Boltzmann constant (SI unit: $W/(m^2 \cdot K^4)$).
- $\beta = \kappa + \sigma_s$ is the extinction coefficient (SI unit: 1/m).
- σ_s is the scattering coefficient (SI unit: 1/m).
- $\phi(\Omega', \Omega)$ is the scattering phase function (dimensionless).
- T is the temperature (SI unit: K).

It takes into account the absorbed, emitted, and scattered radiation of the participating medium.

Two approximation methods are available for the radiation discretization method. The characteristics of each of them are summarized in the following table.

TABLE 6-2: DISCRETIZATION METHODS FOR RADIATION IN PARTICIPATING MEDIA (RPM INTERFACE)

OPTION	DISCRETE ORDINATES METHOD	P1 APPROXIMATION
Optical thickness validity	All	$\tau >> 1$
Scattering modeling	Linear Polynomial	Linear
Computation of G	$G \approx \sum_{i=1}^{N} \omega_i I_i$	$\nabla \cdot (D_{P1} \nabla G) + \kappa(G - 4\pi I_b) = 0$
Computational cost	High: up to 80 additional degrees of freedom (I_i)	Medium: 1 additional degree of freedom (G)

MODELS INPUTS

There is one standard model input — the **Temperature** T. The default is 293.15 K and is used in the blackbody radiative intensity expression.

ABSORPTION

The **Absorption coefficient** κ should be specified. It defines the amount of radiation, $\kappa I(\Omega)$, that is absorbed by the medium.

SCATTERING

This section sets the scattering property of the participating medium.

The **Scattering coefficient** σ_s should be specified.

Choose in addition the **Scattering type**: **Isotropic**, **Linear anisotropic**, or **Polynomial anisotropic** (only with **Discrete ordinates method**). This setting provides options to approximate the scattering phase function using the cosine of the scattering angle, μ_0:

- **Isotropic** (the default) corresponds to the scattering phase function $\phi(\mu_0) = 1$.
- For **Linear anisotropic** it defines the scattering phase function as $\phi(\mu_0) = 1 + a_1\mu_0$. Enter the **Legendre coefficient** a_1.
- For **Polynomial anisotropic** it defines the scattering phase function as

$$\phi(\mu_0) = 1 + \sum_{m=1}^{12} a_m P_m(\mu_0)$$

  Enter each **Legendre coefficient** $a_1, \ldots, a_{12}$ as required.

INITIAL VALUES

When **Discrete ordinates method** is selected as the **Radiation discretization method** for the physics interface, the **Initial radiative intensity** I should be specified. The default is `rpm.Ibinit`, which is the blackbody radiative intensity at initial temperature.

When **P1 approximation** is selected as the **Radiation discretization method** for the physics interface, the **Initial incident radiation** G should be specified. The default is

`(4*pi)*rpm.Ibinit`, computed from the blackbody radiative intensity at initial temperature.

Theory for Radiation in Participating Media

Discrete Ordinates Method (DOM)

P1 Approximation Theory

Radiative Heat Transfer in a Utility Boiler: Application Library path **Heat_Transfer_Module/Thermal_Radiation/boiler**

Radiative Heat Transfer in Finite Cylindrical Media: Application Library path **Heat_Transfer_Module/Verification_Examples/cylinder_participating_media**

Radiative Heat Transfer in Finite Cylindrical Media—P1 Method: Application Library path **Heat_Transfer_Module/Verification_Examples/cylinder_participating_media_p1**

LOCATION IN USER INTERFACE

Context menus

Radiation in Participating Media>Radiation in Participating Media

Ribbon

Physics Tab with **Radiation in Participating Media** selected:

Domains>Radiation in Participating Media>Radiation in Participating Media

Solid

This node uses this version of the heat equation to model heat transfer in solids:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \qquad (6\text{-}9)$$
$$\mathbf{q} = -k\nabla T$$

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the solid density.
- C_p (SI unit: J/(kg·K)) is the solid heat capacity at constant pressure.

- k (SI unit: W/(m·K)) is the solid thermal conductivity (a scalar or a tensor if the thermal conductivity is anisotropic).
- $\mathbf{u}$ (SI unit: m/s) is the velocity field defined by the Translational Motion subnode when parts of the model are moving in the material frame.
- Q (SI unit: W/m^3) is the heat source (or sink). Add one or several heat sources as separate physics features. See Heat Source node and Thermoelastic Damping subnode for example.

For a steady-state problem the temperature does not change with time and the first term disappears.

SOLID MATERIAL

This section is available only when the **Local Thermal Non-Equilibrium** multiphysics coupling is included in the component to model porous media. It makes it possible to define different material properties for the porous matrix and the fluid.

Select any material from the list to define the **Solid material**. The default uses the **Domain material**.

HEAT CONDUCTION, SOLID

The thermal conductivity k describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k\nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic** enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

The components of the thermal conductivity k when given on tensor form (k_{xx}, k_{yy}, and so on, representing an anisotropic thermal conductivity) are available as `ht.kxx`, `ht.kyy`, and so on (using the default name `ht`). The single scalar mean effective thermal conductivity `ht.kmean` is the mean value of the diagonal elements k_{xx}, k_{yy}, and k_{zz}.

Fourier's law assumes that the thermal conductivity tensor is symmetric. A nonsymmetric tensor can lead to unphysical results.

THERMODYNAMICS, SOLID

This section sets the thermodynamics properties of the solid.

The heat capacity at constant pressure describes the amount of heat energy required to produce a unit temperature change in a unit mass.

The **Density** ρ and **Heat capacity at constant pressure** C_p should be specified.

In addition, the thermal diffusivity α, defined as $k/(\rho C_p)$ (SI unit: m^2/s), is also a predefined quantity. The thermal diffusivity can be interpreted as a measure of thermal inertia (heat propagates slowly where the thermal diffusivity is low, for example). The components of the thermal diffusivity α, when given on tensor form (α_{xx}, α_{yy}, and so on, representing an anisotropic thermal diffusivity) are available as `ht.alphaTdxx`, `ht.alphaTdyy`, and so on (using the default physics name `ht`). The single scalar mean thermal diffusivity `ht.alphaTdMean` is the mean value of the diagonal elements α_{xx}, α_{yy}, and α_{zz}. The denominator ρC_p is the effective volumetric heat capacity which is also available as a predefined quantity, `ht.C_eff`.

	Local Thermal Non-Equilibrium Theory for Heat Transfer in Solids

	The Thermoelastic Damping subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

	When **Surface-to-surface radiation** is activated, the Opacity subnode is automatically added to the entire selection, with **Opaque** option selected. The domain selection can't be edited. To set some part of the domain as transparent, add a new Opacity subnode from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

	Heat Generation in a Disc Brake: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc**

	For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Solid
Heat Transfer in Solids>Solid
Heat Transfer in Fluids>Solid
Heat Transfer in Porous Media>Solid
Heat Transfer in Building Materials>Solid
Bioheat Transfer>Solid
Heat Transfer with Surface-to-Surface Radiation>Solid
Heat Transfer with Radiation in Participating Media>Solid

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Domains>*interface***>Solid**

Thermal Dispersion

This subnode should be used to model the heat transfer due to hydrodynamic mixing in a fluid flowing through a porous medium. It adds an extra term $\nabla \cdot (k_{\mathrm{disp}}\nabla T)$ to the right-hand side of the heat equation in porous media, through the modification of the effective thermal conductivity k_{eff} with the dispersive thermal conductivity k_{disp}:

$$(\rho C_p)_{\mathrm{eff}}\frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T = \nabla \cdot (k_{\mathrm{eff}}\nabla T) + Q$$

and specifies values for the longitudinal and transverse dispersivities.

DISPERSIVITIES

This section sets the **Longitudinal dispersivity** λ_{lo} and **Transverse dispersivity** λ_{tr} used for the definition of the tensor of dispersive thermal conductivity:

$$(k_{\mathrm{disp}})_{ij} = \rho_{\mathrm{L}} C_{p,\,\mathrm{L}} D_{ij}$$

where D_{ij} is the dispersion tensor

$$D_{ij} = \lambda_{ijkl}\frac{u_k u_l}{|\mathbf{u}|}$$

and λ_{ijkl} is the fourth-order dispersivity tensor

$$\lambda_{ijkl} = \lambda_{tr}\delta_{ij}\delta_{kl} + \frac{\lambda_{lo} - \lambda_{tr}}{2}(\delta_{ik}\delta_{jl} + \delta_{il}\delta_{jk})$$

If the **Extended** porous matrix model is selected under **Physical Model** (with the Subsurface Flow Module), the **Transverse horizontal dispersivity** $\lambda_{tr,\,h}$ and **Transverse vertical dispersivity** $\lambda_{tr,\,v}$ are defined instead of the **Transverse dispersivity** λ_{tr}.

In this case it is assumed that z is the vertical direction and it defines the dispersion tensor as

$$D_{ij} =$$

$$\frac{1}{|\mathbf{u}|}\begin{bmatrix} \lambda_{lo}u^2 + \lambda_{tr,\,h}v^2 + \lambda_{tr,\,v}w^2 & (\lambda_{lo} - \lambda_{tr,\,h})uv & (\lambda_{lo} - \lambda_{tr,\,v})uw \\ (\lambda_{lo} - \lambda_{tr,\,h})uv & \lambda_{tr,\,h}u^2 + \lambda_{lo}v^2 + \lambda_{tr,\,v}w^2 & (\lambda_{lo} - \lambda_{tr,\,v})vw \\ (\lambda_{lo} - \lambda_{tr,\,v})uw & (\lambda_{lo} - \lambda_{tr,\,v})vw & \lambda_{tr,\,v}u^2 + \lambda_{tr,\,v}v^2 + \lambda_{lo}w^2 \end{bmatrix}$$

The former formulation corresponds to the general formulation when $\lambda_{tr,\,h} = \lambda_{tr,\,v}$.

Porous Medium

The **Thermal Dispersion** node is only available with certain COMSOL products. For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Porous Media>Porous Medium>Thermal Dispersion

More locations are available if the **Heat transfer in porous media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Porous Medium>Thermal Dispersion

Ribbon

Physics Tab with **Porous Medium** selected in the model tree:

Attributes>Thermal Dispersion

Thermoelastic Damping

This subnode should be used to model heat generation due to changes in stress, which may be important in small structures vibrating at high frequencies.

THERMOELASTIC DAMPING

The **Thermoelastic damping** Q_{ted} should be specified either as a **User defined** value, or as the thermoelastic damping contribution straight from the solid mechanics interfaces, when you add a **Thermal Expansion** subnode (with the Structural Mechanics Module). In the latter case it is defined by

$$Q_{\mathrm{ted}} = -\alpha T : \frac{\partial S}{\partial t}$$

where S is the second Piola-Kirchhoff tensor and α is the coefficient of thermal expansion.

Solid

Thermal Expansion (for materials) in the *Structural Mechanics Module User's Guide*

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Solid>Thermoelastic Damping

More locations are available. For example:
Heat Transfer in Fluids>Solid>Thermoelastic Damping

Ribbon

Physics Tab with **Solid** selected in the model tree:
Attributes>Thermoelastic Damping

Translational Motion

This subnode provides movement by translation to the model for heat transfer in solids. It adds the following contribution to the right-hand side of Equation 6-9, defined in the parent node:

$$-\rho C_p \mathbf{u}_{\mathrm{trans}} \cdot \nabla T$$

The contribution describes the effect of a moving coordinate system, which is required to model, for example, a moving heat source.

Special care must be taken on boundaries where $\mathbf{n} \cdot \mathbf{u}_{\mathrm{trans}} \neq 0$. The Heat Flux boundary condition does not, for example, work on boundaries where $\mathbf{n} \cdot \mathbf{u}_{\mathrm{trans}} < 0$.

DOMAIN SELECTION

By default, the selection is the same as for the **Solid** node that it is attached to, but it is possible to use more than one **Translational Motion** subnode, each covering a subset of the **Solid** node's selection.

TRANSLATIONAL MOTION

The x, y, and z (in 3D) components of the **Velocity field** $\mathbf{u}_{\mathrm{trans}}$ should be specified in this section.

Solid

Heat Generation in a Disc Brake: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc**

Friction Stir Welding of an Aluminum Plate: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/friction_stir_welding**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Solid>Translational Motion
Bioheat Transfer>Biological Tissue>Translational Motion

More locations are available. For example:
Heat Transfer in Fluids>Solid>Translational Motion

Ribbon

Physics Tab with **Solid** or **Biological Tissue** selected in the model tree:
Attributes>Translational Motion

Viscous Dissipation

This subnode should be used to account for the heat source coming from the transformation of kinetic energy into internal energy due to viscous stresses. Such effect is expected in fluid regions with large velocity gradients or with high turbulence levels.

VISCOUS DISSIPATION

The Q_{vd} input should be specified either as a **User defined** value, or as the viscous dissipation term contribution straight from the fluid flow interfaces.

For laminar flows, it is defined by

$$Q_{vd} = \tau : \nabla \mathbf{u}$$

where τ is the viscous tensor.

When the **Non-Isothermal Flow** multiphysics coupling node is added, the effect of viscous dissipation can be taken into account by selecting the **Include viscous dissipation** check box under the **Flow Heating** section. In this case, the **Viscous Dissipation** feature is overriden by the multiphysics coupling node's contribution.

Fluid

Porous Medium

Phase Change Material

This feature was previously called Viscous Heating.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Fluids>Fluid>Viscous Dissipation
Heat Transfer in Fluids>Phase Change Material>Viscous Dissipation
Heat Transfer in Fluids>Porous Medium>Viscous Dissipation

More locations are available. For example:
Heat Transfer in Solids>Fluid>Viscous Dissipation
Heat Transfer in Solids>Phase Change Material>Viscous Dissipation
Heat Transfer in Solids>Porous Medium>Viscous Dissipation

Ribbon

Physics Tab with **Fluid**, **Phase Change Material**, or **Porous Medium** selected in the model tree:

Attributes>Viscous Dissipation

Boundary Features

The Heat Transfer interfaces have the following boundary nodes and subnodes available:

- Boundary Heat Source
- Change Thickness (Heat Transfer in Thin Shells Interface)
- Continuity
- Continuity on Interior Boundary
- Deposited Beam Power
- Diffuse Mirror
- Diffuse Surface
- External Temperature (Thin Layer)
- Fracture
- Heat Flux
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Fracture)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Heat Source (Thin Film)
- Incident Intensity
- Inflow Heat Flux
- Initial Values (Heat Transfer in Thin Shells Interface)
- Isothermal Domain Interface
- Layer Heat Source (Thin Layer)
- Line Heat Source on Axis
- Opaque Surface
- Open Boundary
- Outflow
- Periodic Condition
- Prescribed Radiosity
- Radiation Group
- Symmetry
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Conductive Layer (Heat Transfer in Thin Shells Interface)
- Thin Film
- Thin Layer
- Thin Layered Shell (Heat Transfer in Thin Shells Interface)

For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries into account and automatically adds an **Axial Symmetry** node that is valid on the axial symmetry boundaries only.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Boundary Heat Source

This node models a heat source (or heat sink) that is embedded in the boundary. When selected as a **Pair Boundary Heat Source**, it also prescribes that the temperature field is continuous across the pair. Note that in this case the source term is applied on the source side.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

BOUNDARY HEAT SOURCE

Click the **General source** (the default) or **Heat rate** button.

- For **General source** enter a value for the boundary heat source Q_b when the default option, **User defined**, is selected.

  A positive Q_b corresponds to heating and a negative Q_b corresponds to cooling. For the general boundary heat source Q_b, there are predefined heat sources available when simulating heat transfer together with electrical or electromagnetic interfaces. Such sources represent, for example, ohmic heating and induction heating.
- For **Heat rate** enter the heat rate P_b. In this case $Q_b = P_b/A$, where A is the total area of the selected boundaries.

FRAME SELECTION

The settings are the same as for the Heat Source node and are described under **Frame Selection**.

SOURCE POSITION

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Select a **Source position** to define a side where the heat source is defined: **Layer** (the default), **Upside**, or **Downside**. This setting has no effect unless the temperature differs from one side of the boundary to the other. Typically when **Boundary Heat Source** contributes with a **Thin Layer** feature.

To define the boundary heat source Q_b as a function of the temperature, use the local temperature variable on the selected boundary, `ht.bhs1.Tvar`, that corresponds to the appropriate variable (upside, downside, or average temperature of a layer, wall temperature with turbulence modeling), depending on the model configurations. See Boundary Wall Temperature for a thorough description of the boundary temperature variables.

Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.

See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

- Handling Frames in Heat Transfer
- About the Heat Transfer Interfaces

Freeze-Drying: Application Library path **Heat_Transfer_Module/Phase_Change/freeze_drying**

When **Line Heat Flux** is applied on a pair, the flux is only applied on the edge adjacent to the source boundary that is in contact with the destination boundary. Consider adding another pair with opposite source and destination boundaries to apply a flux on the edge adjacent to the destination boundary and in contact with the source boundary.

See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

When **Line Heat Flux** is applied on a pair, the flux is only applied on the edge adjacent to the source boundary which is in contact with the destination boundary. Consider adding another pair with opposite source and destination boundaries to apply a flux on the edge adjacent to the destination boundary and in contact with the source boundary.

See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Boundary Heat Source
Heat Transfer in Solids>Boundary Heat Source
Heat Transfer in Fluids>Boundary Heat Source
Heat Transfer in Porous Media>Boundary Heat Source
Heat Transfer in Building Materials>Boundary Heat Source
Bioheat Transfer>Boundary Heat Source
Heat Transfer with Surface-to-Surface Radiation>Boundary Heat Source
Heat Transfer with Radiation in Participating Media>Boundary Heat Source

Heat Transfer in Solids>Pairs>Pair Boundary Heat Source
Heat Transfer in Fluids>Pairs>Pair Boundary Heat Source
Heat Transfer in Porous Media>Pairs>Pair Boundary Heat Source
Heat Transfer in Building Materials>Pairs>Pair Boundary Heat Source
Bioheat Transfer>Pairs>Pair Boundary Heat Source
Heat Transfer with Surface-to-Surface Radiation>Pairs>Pair Boundary Heat Source
Heat Transfer with Radiation in Participating Media>Pairs>Pair Boundary Heat Source

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Boundary Heat Source**
Pairs>*interface***>Pair Boundary Heat Source**

Change Thickness (Heat Transfer in Thin Shells Interface)

Use this node to give parts of the shell a different thickness than that what is specified on the Heat Transfer in Thin Shells interface **Shell Thickness** section.

CHANGE THICKNESS

Specify a value for the **Shell thickness** d_s. The default value is 0.01 m. This value replaces the overall thickness for the boundaries that are selected.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Change Thickness

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

Boundaries>Heat Transfer in Thin Shells>Change Thickness

Continuity

This node can be added to pairs. It prescribes that the temperature field is continuous across the pair. **Continuity** is only suitable for pairs where the boundaries match.

PAIR SELECTION

Choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

Thermo-Mechanical Analysis of a Surface-Mounted Resistor: Application Library path **Heat_Transfer_Module/Thermal_Stress/surface_resistor**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Pairs>Continuity
Heat Transfer in Solids>Pairs>Continuity
Heat Transfer in Fluids>Pairs>Continuity
Heat Transfer in Porous Media>Pairs>Continuity
Bioheat Transfer>Pairs>Continuity
Heat Transfer in Thin Shells>Pairs>Continuity
Heat Transfer with Surface-to-Surface Radiation>Pairs>Continuity
Heat Transfer with Radiation in Participating Media>Pairs>Continuity
Surface-to-Surface Radiation>Pairs>Continuity
Radiation in Participating Media>Pairs>Continuity

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer in Thin Shells**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media**, **Radiation in Participating Media** or **Surface-to-Surface Radiation** selected:

Pairs>*interface***>Continuity**

Continuity on Interior Boundary

When **Discrete ordinates method** is selected, this node enables intensity conservation across internal boundaries. It is the default boundary condition for all interior boundaries. For **P1 approximation** and **Rosseland approximation** this boundary condition is not available since it is not needed.

Radiative Heat Transfer in a Utility Boiler: Application Library path **Heat_Transfer_Module/Thermal_Radiation/boiler**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Radiation in Participating Media>Radiation in Participating Media>Continuity on Interior Boundary

Radiation in Participating Media>Continuity on Interior Boundary

More locations are available if the **Radiation in participating media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Radiation in Participating Media>Continuity on Interior Boundary

Ribbon

Physics Tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation**, **Heat Transfer with Radiation in Participating Media** or **Radiation in Participating Media** selected:

Boundaries>Radiation in Participating Media>Continuity on Interior Boundary

Deposited Beam Power

This node models heat sources brought by narrow beams, such as laser or electron beams, to a given boundary. This feature is only available in 3D components.

DEPOSITED BEAM POWER

Enter a value for the **Deposited beam power** P_l, the **Beam origin point** $\mathbf{O}$, and the **Beam orientation** $\mathbf{e}$. The orientation vector needs not to be normalized.

BEAM PROFILE

Select a **Distribution type**: **Gaussian** (the default) or **Top-hat disk**.

- For **Gaussian**, enter the **Standard deviation** σ.
- For **Top-hat disk**, enter the **Beam radius** R. Smoothing can be applied by entering a positive **Size of transition zone** ΔR. The default value of 0 m corresponds to an ideal discontinuous top-hat profile.

SOURCE POSITION

To display this section click the **Show** button ( ) and select **Advanced Physics Options**.

Select a **Source position** to define a side where the heat source is defined: **Layer** (the default), **Upside**, or **Downside**. This setting has no effect unless the temperature differs from one side of the boundary to the other.

The standard deviation of the Gaussian distribution and the radius of the top-hat distribution can take values smaller than the mesh element size. For more accurate results, it is important to refine the mesh enough at the deposited heat source location. Otherwise, an automatic substitution replaces the standard deviation σ or the beam radius R by the minimum length to get acceptable results which depends on the mesh element size.

Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.

See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Deposited Beam Power
Heat Transfer in Solids>Deposited Beam Power

Heat Transfer in Fluids>Deposited Beam Power
Heat Transfer in Porous Media>Deposited Beam Power
Bioheat Transfer>Deposited Beam Power
Heat Transfer with Surface-to-Surface Radiation>Deposited Beam Power
Heat Transfer with Radiation in Participating Media>Deposited Beam Power

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Deposited Beam Power**

Diffuse Mirror

This node is a variant of the Diffuse Surface node with a surface emissivity equal to zero. Diffuse mirror surfaces are common as approximations of a surface that is well insulated on one side and for which convection effects can be neglected on the opposite (radiating) side. It resembles a mirror that absorbs all irradiation and then radiates it back in all directions. The node adds radiosity shape function for each spectral band to its selection and uses it as surface radiosity.

The radiative heat flux on a diffuse mirror boundary is zero.

MODEL INPUTS

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups have been added, the model inputs are included here.

There is one standard model input — the **Temperature** T. The default is the temperature variable in the Heat Transfer interface or 293.15 K in the Surface-to-Surface Radiation interface. It is used in the blackbody radiation intensity expression.

RADIATION SETTINGS

Select the **Radiation direction**: **Opacity controlled** (the default), **Negative normal direction**, **Positive normal direction**, or **Both sides**. For information about these options, see Diffuse Surface.

AMBIENT

These settings are the same as for the Diffuse Surface node.

INITIAL VALUES

These settings are the same as for the Diffuse Surface node.

	If this feature is combined with heat transfer in 2D and 1D, the thickness is assumed to be infinite for the view factor computation. The user-defined value for d is still used in the heat transfer equation.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Surface-to-Surface Radiation>Radiation>Diffuse Mirror
Surface-to-Surface Radiation>Diffuse Mirror

More locations are available if the **Surface-to-surface radiation** check box is selected under the **Physical Model** section. For example:
Heat Transfer in Solids>Radiation>Diffuse Mirror

Ribbon

Physics Tab with **Surface-to-Surface Radiation** selected:
Boundaries>Surface-to-Surface Radiation>Diffuse Mirror

Physics Tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:
Boundaries>Radiation>Diffuse Mirror

Diffuse Surface

Diffuse surfaces reflect radiative intensity uniformly in all directions. This node handles radiation with a view factor calculation. The feature adds one radiosity shape function per spectral interval to its selection and uses it as surface radiosity.

It adds a radiative heat source contribution

$$q = \varepsilon(G - e_\mathrm{b}(T))$$

on the side of the boundary where the radiation is defined, where ε is the surface emissivity, G is the irradiation, and $e_\mathrm{b}(T)$ is the blackbody hemispherical total emissive power. Where the radiation is defined on both sides, the radiative heat source is defined on both sides too.

MODEL INPUTS

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups have been added, the model inputs are included here.

There is one standard model input — the **Temperature** T. The default is the temperature variable in the Heat Transfer interface or 293.15 K in the Surface-to-Surface Radiation interface. This model input is used in the expression for the blackbody radiation intensity and, when multiple wavelength intervals are used, for the fractional emissive power. The temperature model input is also used to determine the variable that receives the radiative heat source. When the model input does not contain a dependent variable, the radiative heat source is ignored.

RADIATION SETTINGS

When **Wavelength dependence of emissivity** is set to **Constant** in the **Radiation Settings** section of the physics interface (available when the **Surface-to-surface radiation** check box is selected), select a **Radiation direction** based on the geometric normal `(nx, ny, nz)`: **Opacity controlled**, **Negative normal direction**, **Positive normal direction**, or **Both sides**.

- **Opacity controlled** requires that each boundary is adjacent to exactly one opaque domain. Opacity is controlled by the Opacity domain subfeature. For external boundaries, the exterior side opacity is controlled by the **Exterior radiation** setting at the interface level. This is the default option when the node is added from any version of the Heat Transfer interface with the **Surface-to-surface radiation** check box is selected.
- Select **Negative normal direction** to specify that the surface radiates in the negative normal direction. An arrow indicates the negative normal direction that corresponds to the direction of the radiation emitted by the surface.
- Select **Positive normal direction** if the surface radiates in the positive normal direction. An arrow indicates the positive normal direction that corresponds to the direction of the radiation emitted by the surface.
- Select **Both sides** if the surface radiates on both sides. This is the default option when the node is added from the Heat Transfer in Thin Shells interface or the Surface-to-Surface Radiation interface.

When **Wavelength dependence of emissivity** is set to **Solar and ambient** or **Multiple spectral bands** in the **Radiation Settings** section of the physics interface (available when the **Surface-to-surface radiation** check box is selected), select a **Radiation direction** for each spectral band: **Opacity controlled**, **Negative normal direction**, **Positive normal direction**, **Both sides**, or **None**. The **Radiation direction** defines the radiation direction for each spectral band similarly as when **Wavelength dependence of emissivity** is **Constant**. Defining a radiation direction for each spectral band makes it possible to build models where the transparency or opacity properties defers between spectral bands.

This is useful for example to represent glass opaque to radiation outside of the 0.3–2.5 μm wavelength range.

None is used when adjacent domains are either both transparent or both opaque for a given spectral band.

When the **Surface-to-surface radiation** check box is not selected or not available, the **Radiation Settings** section can be displayed by clicking the **Show** button ( ) and selecting **Advanced Physics Options**. Select a **Radiation direction** between **Negative normal direction** and **Positive normal direction**.

The **Thin Layer** boundary also defines the layer opacity that determines the side of the layer where the radiation occurs, depending on radiation direction. When the **Surface-to-surface radiation** check box is not selected or not available, the thin layer is set opaque.

AMBIENT

If the **Surface-to-surface radiation** check box is selected, select **Define ambient temperature on each side** when the ambient temperature differs between the sides of a boundary. This is needed to define ambient temperature for a surface that radiates on both sides and that is exposed to a hot temperature on one side (for example, fire) and to a cold temperature on the other side (for example, external temperature). By default, **Define ambient temperature on each side** is not selected when the node is added from any version of the Heat Transfer interface; but it is selected when the node is added from the Heat Transfer in Thin Shells interface or the Surface-to-Surface Radiation interface.

Set the **Ambient temperature** T_{amb}. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface. When **Define ambient temperature on each side** is selected, define the **Ambient temperature** $T_{\mathrm{amb,\,u}}$ and $T_{\mathrm{amb,\,d}}$ on the up and down side, respectively. The geometric normal points from the down side to the up side.

Set T_{amb} to the far-away temperature in directions where no other boundaries obstruct the view. Inside a closed cavity, the ambient view factor, F_{amb}, is theoretically zero and the value of T_{amb} therefore should not matter. It is, however, good practice to set T_{amb} to T or to a typical temperature value for the cavity surfaces in such cases because that minimizes errors introduced by the finite resolution of the view factor evaluation.

SURFACE FRACTIONAL EMISSIVE POWER

This section is only available when the **Surface-to-surface radiation** check box is selected.

This section is available when the **Wavelength dependence of emissivity** is defined as **Solar and ambient** or **Multiple spectral bands** for the physics interface (see Radiation Settings).

When the **Fractional emissive power** is **Blackbody/Graybody**, the fractional emissive power is automatically computed for each spectral band as a function of the band endpoints and surface temperature.

When the **Fractional emissive power** is **User defined**, define the **Fractional emissive power**, FEP_{Bi} for each spectral band. All fractional emissive powers are expected to be in [0,1] and their sum is expected to be equal to 1.

SURFACE EMISSIVITY

In diffuse gray and diffuse spectral radiation models, the surface emissivity and the absorptivity must be equal. For this reason it is equivalent to define the surface emissivity or the absorptivity.

The surface emissivity settings are defined per spectral interval.

When the **Radiation direction** is **Opacity controlled**, **Negative normal direction**, or **Positive normal direction** for a spectral band, by default, the **Surface emissivity** ε (dimensionless) uses values **From material**. This is a property of the material surface that depends both on the material itself and the structure of the surface. Make sure that a material is defined at the boundary level (by default materials are defined at the domain level).

When the **Radiation direction** is set to **Both sides** for a spectral band, define the **Material on upside** and **Material on downside**:

- The defaults for both **Material on upside** and **Material on downside** use **Boundary material**. The list has options based on the materials defined in the model.
- Define the **Surface emissivity** on the upside and downside, respectively. The geometric normal points from the down side to the up side. Set the surface emissivity to a number between 0 and 1, where 0 represents diffuse mirror and 1 is appropriate for a perfect blackbody. The proper value for a physical material lies somewhere in-between and can be found from tables or measurements.

When the **Radiation direction** is set to **None** for a spectral band, no information is needed for this spectral band in the **Surface Emissivity** section.

INITIAL VALUES

This section is only available when the **Surface-to-surface radiation** check box is selected.

The surface radiosity initial values are defined per spectral interval.

When the **Radiation direction** is **Opacity controlled**, **Negative normal direction**, or **Positive normal direction** for a spectral band B_i, the default **Surface radiosity** $J_{Bi,\,\mathrm{init}}$ is defined as

$$J_{\mathrm{B}i,\,\mathrm{init}} = \varepsilon_{Bi} e_{\mathrm{b}}(T_{\mathrm{init}}) + (1-\varepsilon_{\mathrm{B}i}) e_{\mathrm{b}}(T_{\mathrm{amb}})$$

When **Both sides** is selected as the **Radiation direction**,

Enter initial values for the **Surface radiosity** $J_{\mathrm{B}i,\,\mathrm{init,\,u}}$ and $J_{\mathrm{B}i,\,\mathrm{init,\,d}}$. The default **Surface radiosity** is `ht.JBiinitU` and `ht.JBiinitD`.

$$J_{\mathrm{B}i,\,\mathrm{init,\,u}} = \varepsilon_{\mathrm{B}i,\,\mathrm{u}} e_{\mathrm{b}}(T_{\mathrm{init}}) + (1-\varepsilon_{\mathrm{B}i,\,\mathrm{u}}) e_{\mathrm{b}}(T_{\mathrm{amb,\,u}})$$

$$J_{\mathrm{B}i,\,\mathrm{init,\,d}} = \varepsilon_{\mathrm{B}i,\,\mathrm{d}} e_{\mathrm{b}}(T_{\mathrm{init}}) + (1-\varepsilon_{\mathrm{B}i,\,\mathrm{d}}) e_{\mathrm{b}}(T_{\mathrm{amb,\,d}})$$

When **None** is selected as the **Radiation direction**, no surface radiosity is defined; hence no initial value is needed.

- In the notation used here, `Bi` stands for `B1`, `B2`,... up to the maximum number of spectral intervals.
- When the model contains one spectral interval, $J_{\mathrm{B}i,\,\mathrm{init}}$, $J_{\mathrm{B}i,\,\mathrm{init,\,u}}$ and $J_{\mathrm{B}i,\,\mathrm{init,\,d}}$ are named, respectively, J_{init}, $J_{\mathrm{init,\,u}}$ and $J_{\mathrm{init,\,d}}$.
- If this feature is combined with heat transfer in 2D and 1D, the thickness is assumed to be infinite for the view factor computation. The user-defined value for d is still used in the heat transfer equation.

To define temperature dependencies for the user inputs (surface emissivity for example), use the temperature variable `ht.T`, that corresponds to the appropriate variable (upside, downside, or average temperature of a layer, wall temperature with turbulence modeling), depending on the model configurations. See Boundary Wall Temperature for a thorough description of the boundary temperature variables.

Several settings for this node depend on the **Wavelength dependence of emissivity** setting, which is defined for the physics interface when the **Surface-to-surface radiation** check box is selected.

In addition, the **Transparent media refractive index** is equal to 1 by default, and can be set when the **Surface-to-surface radiation** check box is selected.

See Radiation Settings.

Theory for Surface-to-Surface Radiation

Upside and downside settings can be visualized by plotting the global normal vector (nx, ny, nz), that always points from downside to upside. Note that the normal vector (ht.nx, ht.ny, ht.nz) may be oriented differently.

See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

Heat Generation in a Disc Brake: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Radiation>Diffuse Surface
Heat Transfer in Solids>Radiation>Diffuse Surface
Heat Transfer in Fluids>Radiation>Diffuse Surface
Heat Transfer in Porous Media>Radiation>Diffuse Surface
Bioheat Transfer>Radiation>Diffuse Surface
Heat Transfer with Surface-to-Surface Radiation>Radiation>Diffuse Surface
Surface-to-Surface Radiation>Diffuse Surface
Heat Transfer with Radiation in Participating Media>Radiation>Diffuse Surface
Heat Transfer in Thin Shells>Radiation>Diffuse Surface

Ribbon

Physics Tab with **Surface-to-Surface Radiation** selected:
Boundaries>Surface-to-Surface Radiation>Diffuse Surface

Physics Tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation**, **Heat Transfer with Radiation in Participating Media**, or **Heat Transfer in Thin Shells** selected:
Boundaries>Radiation>Diffuse Surface

External Temperature (Thin Layer)

Use this subnode to specify the temperature on the exterior side of a layer. This feature is only applicable when **Layer type** is set to **Thermally thick approximation** or **General**, on exterior boundaries.

TEMPERATURE

For **User defined**, enter a value or expression for the **Temperature** T_0. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

The equation for this condition is $T = T_0$ where T_0 is the external temperature to be prescribed.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

	Thin Layer Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Layer>External Temperature

More locations are available. For example:
Heat Transfer in Fluids>Thin Layer>External Temperature

Ribbon

Physics Tab with **Thin Layer** selected in the model tree:
Attributes>External Temperature

Fracture

Use this node to allow heat transfer in fractures inside domains. It can also be used to allow heat transfer in films.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

FRACTURE PROPERTIES

This section is only available when the node is added from one of the versions of the Heat Transfer interface (`ht`). Specify a value for the **Fracture thickness** d_{fr}. The default is 0.01 m.

FLUID MATERIAL

By default, the **Boundary material** is used.

POROUS MATERIAL

By default, the **Boundary material** is used. The **Volume fraction** θ_{fr} should be specified. The default is 0.

HEAT CONDUCTION, FRACTURE

The default **Thermal conductivity** k_{fr} is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity, and enter another value or expression. Select an **Effective conductivity**: **Volume average** (the default) or **Power law**.

THERMODYNAMICS, FRACTURE

The default **Density** ρ_{fr} and **Specific heat capacity** $C_{p,\,\mathrm{fr}}$ are taken **From material**. For **User defined** enter other values or expressions.

The settings for the **Model Inputs**, **Heat Conduction, Fluid**, and **Thermodynamics, Fluid** sections are the same as for Fluid.

Fracture in Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Porous Media>Fracture
Heat Transfer in Porous Media>Pairs>Fracture
Heat Transfer in Fractures>Fracture

More locations are available if the **Heat transfer in porous media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Fracture

Heat Transfer in Solids>Pairs>Fracture

Heat Transfer in Thin Shells>Fracture

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**,

Heat Transfer with Surface-to-Surface Radiation or **Heat Transfer with Radiation in Participating Media** selected:
Boundaries>*interface***>Fracture**
Pairs>*interface***>Fracture**

Physics Tab with *interface* as **Heat Transfer in Thin Shells**, **Heat Transfer in Thin Films** or **Heat Transfer in Fractures** selected:
Boundaries>*interface***>Fracture**

Heat Flux

Use this node to add heat flux across boundaries. A positive heat flux adds heat to the domain. This feature is not applicable to inlet boundaries, use the Inflow Heat Flux condition instead.

HEAT FLUX

Click to select the **General inward heat flux** (the default), **Convective heat flux**, or **Heat rate** button.

General Inward Heat Flux

It adds q_0 to the total flux across the selected boundaries. Enter a value for q_0 to represent a heat flux that enters the domain. For example, any electric heater is well represented by this condition, and its geometry can be omitted.

Convective Heat Flux

The default option is to enter a **User defined** value for the **Heat transfer coefficient** h. In addition, the following options are also available to control the type of convective heat flux to model: **External natural convection**, **Internal natural convection**, **External forced convection**, or **Internal forced convection**.

For all options except **User defined**, select a **Fluid**: **Air** (the default), **Transformer oil**, or **Water**.

For **Air**, also set the **Absolute pressure**, p_A. For **User Defined**, enter a value or expression. The default is 1 atm. Else, select an **Ambient absolute pressure** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

External Natural Convection

For **External natural convection** select **Vertical wall**, **Inclined wall**, **Horizontal plate, upside**, **Horizontal plate, downside**, **Long horizontal cylinder**, **Sphere**, or **Vertical Thin Cylinder** from the list under **Heat transfer coefficient**. Then enter the applicable information:

- **Wall height** L
- **Wall height** L and the **Tilt angle** ϕ. The tilt angle is the angle between the wall and the vertical direction, $\phi = 0$ for vertical walls.
- **Plate diameter (area/perimeter)** L. L is approximated by the ratio between the surface area and its perimeter.
- **Cylinder diameter** D
- **Sphere diameter** D
- **Cylinder height** H

Internal Natural Convection

For **Internal natural convection** select **Narrow chimney, parallel plates** or **Narrow chimney, circular tube** from the list under **Heat transfer coefficient**. Then enter the applicable information:

- **Plate distance** L and a **Chimney height** H.
- **Tube diameter** D and a **Chimney height** H.

External Forced Convection

For **External forced convection** select **Plate, averaged transfer coefficient** or **Plate, local transfer coefficient** from the list under **Heat transfer coefficient**. Then enter the applicable information:

- **Plate length** L and **Velocity, fluid** U.
- **Position along the plate** x_{pl} and **Velocity, fluid** U.

Internal Forced Convection

For **Internal forced convectio**n the only option is **Isothermal tube**. Enter a **Tube diameter** D and a **Velocity, fluid** U.

If **Velocity, fluid** U is **User defined**, enter a value or expression. Else, select a **Wind velocity** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

Finally, enter an **External temperature**, T_{ext}. The value depends on the geometry and the ambient flow conditions. Convective heat flux is defined by $q_0 = h(T_{\mathrm{ext}} - T)$. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

Heat Rate

For **Heat rate** enter the heat rate P_0 across the boundaries where the **Heat Flux** node is active. In this case $q_0 = P_0/A$, where A is the total area of the selected boundaries.

FRAME SELECTION

The settings are the same as for the Heat Source node and are described under **Frame Selection**.

The detailed definition of the predefined heat transfer coefficients is given in The Heat Transfer Coefficients.

For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 16.

- Handling Frames in Heat Transfer
- About the Heat Transfer Interfaces

- *Power Transistor*: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/power_transistor**
- *Free Convection in a Water Glass*: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/cold_water_glass**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Heat Flux
Heat Transfer in Solids>Heat Flux
Heat Transfer in Fluids>Heat Flux
Heat Transfer in Porous Media>Heat Flux
Bioheat Transfer>Heat Flux
Heat Transfer with Surface-to-Surface Radiation>Heat Flux
Heat Transfer with Radiation in Participating Media>Heat Flux

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**,

Heat Transfer with Surface-to-Surface Radiation or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Heat Flux**

Heat Flux (Heat Transfer in Thin Shells Interface)

This node adds a heat flux $q_{0,\,\mathrm{u}}$ for the upside heat flux and a heat flux $q_{0,\,\mathrm{d}}$ for the downward heat flux to the right-hand side of Equation 6-4

$$d_{\mathrm{s}}\rho C_p\frac{\partial T}{\partial t}-\nabla\cdot(d_{\mathrm{s}}k\nabla T) = d_{\mathrm{s}}Q+d_{\mathrm{s}}q_0 \tag{6-10}$$

where

$$q_0 = q_{0,\,\mathrm{u}}+q_{0,\,\mathrm{d}}$$

For the convective heat flux, q_0 is defined as:

$$q_0 = h_{\mathrm{u}}(T_{\mathrm{ext,\,u}}-T)+h_{\mathrm{d}}(T_{\mathrm{ext,\,d}}-T)$$

FRAME SELECTION

To display this section add both a **Heat Transfer in Thin Shells (htsh)** and **Moving Mesh (ale)** interface (found under the **Mathematics>Deformed Mesh** branch when adding a physics interface). Then click the **Show** button ( ) and select **Advanced Physics Options**.

The rest of the settings are the same as for the Out-of-Plane Heat Flux node as described under **Frame Selection** for About the Heat Transfer Interfaces.

UPSIDE CONVECTIVE HEAT FLUX

Select between specifying the upside convective heat flux directly or as a convective term using a heat transfer coefficient.

General Inward Heat Flux

General inward heat flux is selected by default. Enter a value or expression for the inward (or outward, if the quantity is negative) heat flux through the upside in the $q_{0,\,\mathrm{u}}$ field.

Convective Heat Flux

Click the **Convective heat flux** button to specify an inward (or outward, if the quantity is negative) heat flux through the upside as $h_{\mathrm{u}}\cdot(T_{\mathrm{ext,\,u}}-T)$.

Select a **Heat transfer coefficient** h_u to control the type of convective heat flux to model: **User defined** (the default), **External natural convection**, **Internal natural convection**, **External forced convection**, or **Internal forced convection**. If convective flux is only required on the downside, use the default, which sets $h_u = 0$.

For all of the options, enter an **External temperature**, $T_{ext, u}$. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

The rest of the settings are the same for the Out-of-Plane Heat Flux node, as described for the Heat Transfer interface under Convective Heat Flux.

DOWNSIDE INWARD HEAT FLUX

The options in the **Downside Inward Heat Flux** section are the same as in the **Upside Inward Heat Flux** section except it applies to the downside instead of the upside.

In 2D the heat flux contribution is multiplied by d_z to account for the out-of-plane thickness.

Theory for Heat Transfer in Thin Structures

The Heat Transfer Coefficients

Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.

See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

Disk-Stack Heat Sink: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/disk_stack_heat_sink**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Heat Flux

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

Boundaries>Heat Transfer in Thin Shells>Heat Flux

Heat Source (Fracture)

Use this subnode to add an internal heat source Q_{fr} within the fracture. Add one or more heat sources. If this subnode is applied on a pair, note that the source term is applied on the source side.

HEAT SOURCE

Select the **General source** (the default) or **Heat rate** button to define Q_{fr}.

- For **General source** enter a value or expression for Q_{fr} as a heat source per volume.
- For **Heat rate** define the heat rate P_{fr}. In this case $Q_{fr} = P_{fr}/V$ where $V = Ad_{fr}$ with A equal to the area of the boundary selection.

Fracture in Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Porous Media>Fracture>Heat Source

Heat Transfer in Porous Media>Pairs>Fracture>Heat Source

More locations are available if the **Heat transfer in porous media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Fracture>Heat Source

Heat Transfer in Solids>Pairs>Fracture>Heat Source

Ribbon

Physics Tab with **Fracture** selected in the model tree:

Attributes>Heat Source

Heat Source (Heat Transfer in Thin Shells Interface)

This node describes heat generation within the shell through a d_sQ contribution to the right-hand side of Equation 4-45. If it is applied on a pair, note that the source term is applied on the source side.

Express heating and cooling with positive and negative values, respectively. Add one or more nodes as needed; all heat sources within a boundary contribute to the total heat source. Specify the heat source as the heat per volume in the domain, as a linear heat source, or as a heat rate.

HEAT SOURCE

These settings are the same as for the Heat Source node available for the other interfaces.

FRAME SELECTION

The settings are the same as for the Heat Source node **Frame Selection** section

In 2D components the heat source is multiplied by d_z to account for the out-of-plane thickness.

Theory for Heat Transfer in Thin Structures

Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Heat Source

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

Boundaries>Heat Transfer in Thin Shells>Heat Source

Heat Source (Thin Film)

Use this subnode to add an internal heat source Q_f within the thin film. Add one or more heat sources. If this subnode is applied on a pair, note that the source term is applied on the source side.

HEAT SOURCE

Select the **General source** (the default) or **Heat rate** button to define Q_f.

- For **General source** enter a value or expression for Q_f as a heat source per volume.
- For **Heat rate** define the heat rate P_f. In this case $Q_f = P_f / V$ where $V = Ad_f$ with A equal to the area of the boundary selection.

Thin Film

Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Film>Heat Source

More locations are available. For example:
Heat Transfer in Fluids>Thin Film>Heat Source

Ribbon

Physics Tab with **Thin Film** selected in the model tree:

Attributes>Heat Source

Incident Intensity

Use this node to specify the radiative intensity along incident directions on a boundary. This intensity is a power per unit solid angle and unit surface area projected onto the plane normal to the radiation direction.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

INCIDENT INTENSITY

The **Boundary radiation intensity** *I* should be specified. This represents the value of radiative intensity along incoming discrete directions. Values of radiative intensity on outgoing discrete directions are not prescribed.

When **Discrete ordinates method** is selected, the components of each discrete ordinate vector can be used in this expression. The syntax is `name.sx`, `name.sy`, and `name.sz`, where `name` is the physics interface node name. By default, the Heat Transfer interface is `ht` so `ht.sx`, `ht.sy`, and `ht.sz` correspond to the components of discrete ordinate vectors.

When **P1 approximation** is selected as the **Radiation discretization method** for the physics interface, there is additional theory, equations, and variables described in P1 Approximation Theory.

Radiation in Participating Media (RPM Interface)

Theory for Radiation in Participating Media

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Radiation in Participating Media>Radiation in Participating Media>Incident Intensity
Radiation in Participating Media>Incident Intensity

Heat Transfer with Radiation in Participating Media>Pairs>Incident Intensity
Radiation in Participating Media>Pairs>Incident Intensity

More locations are available if the **Radiation in participating media** check box is selected under the **Physical Model** section. For example:
Heat Transfer in Solids>Radiation in Participating Media>Incident Intensity

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation**, **Heat Transfer with Radiation in Participating Media** or **Radiation in Participating Media** selected:
Boundaries>Radiation in Participating Media>Incident Intensity
Pairs>*interface***>Incident Intensity**

Inflow Heat Flux

Use this node to model inflow of heat through a virtual domain with a heat source. The temperature at the outer boundary of the virtual domain is known. This boundary condition estimates the heat flux through the system boundary

$$-\mathbf{n}\cdot\mathbf{q} = -q_0\frac{A(\mathbf{u}\cdot\mathbf{n})}{\int_S|\mathbf{u}\cdot\mathbf{n}|ds}+\rho(h_{\mathrm{in}}-h_{\mathrm{ext}})\mathbf{u}\cdot\mathbf{n} \tag{6-11}$$

where A is the total area of the selected boundaries and

$$h_{\mathrm{in}}-h_{\mathrm{ext}} = \int_{T_{\mathrm{ext}}}^{T_{\mathrm{in}}}C_p dT+\int_{p_{\mathrm{ext}}}^{p_{\mathrm{A}}}\frac{1}{\rho}(1-\alpha_p)dp \tag{6-12}$$

A positive heat flux adds heat to the domain. This feature is applicable to inlet boundaries of non-solid domains.

INFLOW HEAT FLUX

Select the **Inward heat flux** (the default) or **Heat rate** buttons.

- For **Inward heat flux** define q_0 to add to the total flux across the selected boundaries.
- For **Heat rate** define the heat rate P_0. In this case $q_0 = P_0/A$.

For either selection, the **External temperature** T_{ext} and the **External absolute pressure** p_{ext} should be specified. For **User Defined**, enter values or expressions. Else, select an **Ambient temperature** and an **Ambient absolute pressure** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Inflow Heat Flux
Heat Transfer in Solids>Inflow Heat Flux
Heat Transfer in Fluids>Inflow Heat Flux
Heat Transfer in Porous Media>Inflow Heat Flux
Bioheat Transfer>Inflow Heat Flux
Heat Transfer with Surface-to-Surface Radiation>Inflow Heat Flux
Heat Transfer with Radiation in Participating Media>Inflow Heat Flux

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**,

Heat Transfer with Surface-to-Surface Radiation or **Heat Transfer with Radiation in Participating Media** selected:
Boundaries>*interface***>Inflow Heat Flux**

Initial Values (Heat Transfer in Thin Shells Interface)

This node adds an initial value for the temperature that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. If more than one set of initial values is needed, add an **Initial Values** node from the **Physics** toolbar.

INITIAL VALUES

For **User defined**, enter a value or expression for the initial value of the **Temperature** T. The default is approximately room temperature, 293.15 K (20° C). Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

Theory for Heat Transfer in Thin Structures

Disk-Stack Heat Sink: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/disk_stack_heat_sink**

LOCATION IN USER INTERFACE

Context menus
Heat Transfer in Thin Shells>Initial Values

Ribbon
Physics Tab with **Heat Transfer in Thin Shells** selected:
Boundaries>Heat Transfer in Thin Shells>Initial Values

Isothermal Domain Interface

This node adds several conditions for temperature exchange at the interfaces of an isothermal domain. It can be applied on interior and exterior boundaries of the computational domain, except on the boundaries found inside a single isothermal domain.

In addition to the default **Isothermal domain Interface** node always present when the **Isothermal domain** check box is selected under **Physical Model** section, you can add more **Isothermal domain Interface** nodes if needed.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

ISOTHERMAL DOMAIN INTERFACE

The available **Interface type** options are **Thermal insulation** (the default), **Continuity**, **Ventilation**, **Convective heat flux**, and **Thermal contact**. Depending on the **Interface type** selected from the list, further settings are required.

The description of all the available options is summarized in the following table:

TABLE 6-3: ISOTHERMAL DOMAIN INTERFACE OPTIONS

OPTION	ISOTHERMAL DOMAIN	ADJACENT DOMAIN	DESCRIPTION
Thermal Insulation	Any	Any	No flux
Continuity	Any	Any	Temperature continuity
Ventilation	Fluid	Fluid	Mass flux
Convective Heat Flux	Solid	Fluid	Heat flux (convective)
Thermal Contact	Solid	Solid	Heat flux (conductive)

Thermal Insulation

This condition should be used if no heat exchange occurs between the isothermal domain and the adjacent domain.

Continuity

With this condition, the temperatures of each side of the boundary are forced to be equal.

Ventilation

This condition specifies the mass flux at the interface between an isothermal domain and another domain containing the same fluid.

The available **Flow direction** options are **Positive normal direction** (the default), **Negative normal direction**, and **Both sides**. For each option, either the **Mass flux** or the **Mass flow rate** should be specified:

- $\varphi_{d \to u}$ is the mass flux in the positive direction and $\varphi_{u \to d}$ is the mass flux in the negative direction.
- $\Phi_{d \to u}$ is the mass flow rate in the positive direction and $\Phi_{u \to d}$ is the mass flow rate in the negative direction.

The **External temperature** found under the section of the same name must be set when the isothermal domain interface is also an exterior boundary.

Convective Heat Flux

This condition specifies the convective heat flux at the interface of a solid isothermal domain adjacent to a fluid.

Either the **Heat transfer coefficient** h or the **Thermal conductance** h_{th} should be specified.

The **External temperature** found under the section of the same name must be set when the isothermal domain interface is also an exterior boundary.

Thermal Contact

This condition specifies the conductive heat flux at the interface of a solid isothermal domain adjacent to a solid.

Either the **Thermal resistance**, R_t or the **Absolute thermal resistance**, $R_{t, th}$ should be specified.

The **External temperature** found under the section of the same name must be set when the isothermal domain interface is also an exterior boundary.

EXTERNAL TEMPERATURE

This section is not available if the **Interface type** is set to **Thermal Insulation** or **Continuity**. Else, the **External temperature**, T_{ext} should be specified. It is used to compute the heat exchange with the exterior by the **Ventilation**, **Convective heat flux**, and **Thermal contact** options on isothermal domain interfaces that are also exterior boundaries. The value is ignored on interior boundaries.

For **User defined**, enter a value or expression for the **Temperature** T_{ext}. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

	Isothermal Domain Theory for Lumped Isothermal Domain

	Natural Convection Cooling of a Vacuum Flask: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/vacuum_flask**

LOCATION IN USER INTERFACE

Context menus

If the **Isothermal domain** check box is selected under the **Physical Model** section:

Heat Transfer>Isothermal Domain Interface

Heat Transfer in Solids>Isothermal Domain Interface

Heat Transfer in Fluids>Isothermal Domain Interface

Heat Transfer in Porous Media>Isothermal Domain Interface

Bioheat Transfer>Isothermal Domain Interface

Heat Transfer with Surface-to-Surface Radiation>Isothermal Domain Interface

Heat Transfer with Radiation in Participating Media>Isothermal Domain Interface

Heat Transfer in Solids>Pairs>Isothermal Domain Interface
Heat Transfer in Fluids>Pairs>Isothermal Domain Interface
Heat Transfer in Porous Media>Pairs>Isothermal Domain Interface
Bioheat Transfer>Pairs>Isothermal Domain Interface
Heat Transfer with Surface-to-Surface Radiation>Pairs>Isothermal Domain Interface
Heat Transfer with Radiation in Participating Media>Pairs>Isothermal Domain Interface

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Isothermal Domain Interface**

Pairs>*interface***>Isothermal Domain Interface**

Layer Heat Source (Thin Layer)

Use this subnode to add an internal heat source Q_s within the thin layer. Add one or more heat sources. If this subnode is applied on a pair, note that the source term is applied on the source side.

LAYER HEAT SOURCE

Select the **General source** (the default) or **Heat rate** button to define Q_s.

- For **General source** enter a value or expression for Q_s as a heat source per volume.
- For **Heat rate** define the heat rate P_s. In this case $Q_s = P_s / V$ where $V = Ad_s$ with A equal to the area of the boundary selection.

	Thin Layer Theory for Heat Transfer in Thin Structures

	The **Heat rate** option is only available when the **Layer type** is **Thermally thin approximation** or **Thermally thick approximation** in the Thin Layer parent node. With **General** layer type, a general source user input, $Q_{s,i}$, is available for each layer. In this case the layers are numbered from the upside (Layer 1) to the downside.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Layer>Layer Heat Source

More locations are available. For example:
Heat Transfer in Fluids>Thin Layer>Layer Heat Source

Ribbon

Physics Tab with **Thin Layer** selected in the model tree:
Attributes>Layer Heat Source

Line Heat Source on Axis

This node, available for 2D axisymmetric components, models a heat source (or sink) that is so thin that it has no thickness in the model geometry. The settings are the same as for the Line Heat Source node.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Line Heat Source on Axis
Heat Transfer in Solids>Line Heat Source on Axis
Heat Transfer in Fluids>Line Heat Source on Axis
Heat Transfer in Porous Media>Line Heat Source on Axis
Bioheat Transfer>Line Heat Source on Axis
Heat Transfer with Surface-to-Surface Radiation>Line Heat Source on Axis
Heat Transfer with Radiation in Participating Media>Line Heat Source on Axis

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:
Boundaries>*interface***>Line Heat Source on Axis**

Opaque Surface

This node defines a boundary opaque to radiation. It prescribes incident intensities on a boundary and accounts for the net radiative heat flux, q_{w}, that is absorbed by the surface.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

MODELS INPUTS

This section has fields and values that are inputs to expressions that define material properties. If such user-defined materials are added, the model inputs appear here.

There is one standard model input — the **Temperature** T, which is used in the blackbody radiative intensity expression.

> The boundary temperature definition can differ from that of the temperature in the adjacent domain.

WALL SETTINGS

Select a **Wall type** to define the behavior of the wall: **Gray wall** or **Black wall**.

Gray Wall

If **Gray wall** is selected the default **Surface emissivity** ε value is taken **From material** (a material defined on the boundaries). For **User defined** enter another value or expression.

An emissivity of 0 means that the surface emits no radiation at all and that all outgoing radiation is diffusely reflected by this boundary. An emissivity of 1 means that the surface is a perfect blackbody: outgoing radiation is fully absorbed on this boundary. The radiative intensity along incoming discrete directions on this boundary is defined by

$$I_{i,\,\mathrm{bnd}} = \varepsilon I_{\mathrm{b}}(T) + \frac{1-\varepsilon}{\pi} q_{\mathrm{out}}$$

Black Wall

If **Black wall** is selected, no user input is required, and the radiative intensity along the incoming discrete directions on this boundary is defined by

$$I_{i,\,\mathrm{bnd}} = I_{\mathrm{b}}(T)$$

Values of radiative intensity along outgoing discrete directions are not prescribed.

When **P1 approximation** is selected as the **Radiation discretization method** for the physics interface, there is additional theory, equations, and variables described in P1 Approximation Theory.

Radiation in Participating Media (RPM Interface)

Theory for Radiation in Participating Media

Radiative Heat Transfer in a Utility Boiler: Application Library path **Heat_Transfer_Module/Thermal_Radiation/boiler**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Radiation in Participating Media>Radiation in Participating Media>Opaque Surface
Radiation in Participating Media>Opaque Surface

Heat Transfer with Radiation in Participating Media>Pairs>Opaque Surface
Radiation in Participating Media>Pairs>Opaque Surface

More locations are available if the **Radiation in participating media** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Radiation in Participating Media>Opaque Surface

Heat Transfer in Solids>Pairs>Opaque Surface

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation**, **Heat Transfer with Radiation in Participating Media** or **Radiation in Participating Media** selected:

Boundaries>Radiation in Participating Media>Opaque Surface

Pairs>*interface***>Opaque Surface**

Open Boundary

This node adds a boundary condition for modeling heat flux across an open boundary; the heat can flow out of the domain or into the domain with a specified exterior temperature. Use this node to limit a modeling domain that extends in an open fashion.

OPEN BOUNDARY

Enter the exterior **Temperature** T_0 outside the open boundary.

Natural Convection Cooling of a Vacuum Flask: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/vacuum_flask**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Open Boundary
Heat Transfer in Solids>Open Boundary
Heat Transfer in Fluids>Open Boundary
Heat Transfer in Porous Media>Open Boundary
Bioheat Transfer>Open Boundary
Heat Transfer with Surface-to-Surface Radiation>Open Boundary
Heat Transfer with Radiation in Participating Media>Open Boundary

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Open Boundary**

Outflow

This node provides a suitable boundary condition for convection-dominated heat transfer at outlet boundaries. In a model with convective heat transfer, this condition states that the only heat transfer occurring across the boundary is by convection. The temperature gradient in the normal direction is zero, and there is no radiation. This is usually a good approximation of the conditions at an outlet boundary in a heat transfer model with fluid flow.

BOUNDARY SELECTION

In most cases, the Outflow node does not require any user input. If required, select the boundaries that are convection-dominated outlet boundaries.

Heat Sink: Application Library path
Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/heat_sink

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Outflow
Heat Transfer in Solids>Outflow
Heat Transfer in Fluids>Outflow

Heat Transfer in Porous Media>Outflow
Bioheat Transfer>Outflow
Heat Transfer with Surface-to-Surface Radiation>Outflow
Heat Transfer with Radiation in Participating Media>Outflow

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Outflow**

Periodic Condition

Use this node to add periodic temperature conditions to boundary pairs. The **Destination Selection** subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

For information about the **Orientation of Source** section, see Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

PERIODIC CONDITION

Enter a **Temperature offset** ΔT to the temperature periodicity. The default value is 0 K, so that the source and destination temperatures are equal.

Convection Cooling of Circuit Boards—3D Forced Convection: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/circuit_board_forced_3d**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Periodic Condition
Heat Transfer in Solids>Periodic Condition
Heat Transfer in Fluids>Periodic Condition
Heat Transfer in Porous Media>Periodic Condition
Bioheat Transfer>Periodic Condition
Heat Transfer with Surface-to-Surface Radiation>Periodic Condition
Heat Transfer with Radiation in Participating Media>Periodic Condition

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Periodic Condition**

Prescribed Radiosity

Use this node to specify radiosity on the boundary for each spectral band. Radiosity can be defined as *blackbody* or *graybody radiation*. A user-defined surface radiosity expression can also be defined.

MODEL INPUTS

These settings are the same as for Diffuse Mirror.

RADIATION DIRECTION

When **Wavelength dependence of emissivity** is set to **Constant** select a **Radiation direction** based on the geometric normal `(nx, ny, nz)`: **Opacity controlled** (the default), **Negative normal direction**, **Positive normal direction**, or **Both sides**.

- **Opacity controlled** requires that each boundary is adjacent to exactly one opaque domain. Opacity is controlled by the Opacity condition.
- Select **Negative normal direction** to specify that the surface radiates in the negative normal direction.
- Select **Positive normal direction** if the surface radiates in the positive normal direction.
- Select **Both sides** if the surface radiates on both sides.

When **Wavelength dependence of emissivity** is set to **Solar and ambient** or **Multiple spectral bands** select a **Radiation direction** for each spectral band: **Opacity controlled** (the default), **Negative normal direction**, **Positive normal direction**, or **Both sides**, or **None**. The **Radiation direction** defines the radiation direction for each spectral band similarly as when **Wavelength dependence of emissivity** is **Constant**.

If this feature is combined with heat transfer in 2D and 1D, the thickness is assumed to be infinite for the view factor computation. The user-defined value for d is still used in the heat transfer equation.

Radiosity does not directly affect the boundary condition on the boundary where it is specified, but rather how that boundary affects others through radiation.

Select a **Radiosity expression**: **Graybody radiation** (the default), **Blackbody radiation**, or **User defined**.

Blackbody Radiation

When **Blackbody radiation** is selected it sets the surface radiosity expression corresponding to a blackbody.

- When **Wavelength dependence of emissivity** is set to **Constant** it defines $J = e_{\mathrm{b}}(T)$ when radiation is defined on one side or $J_{\mathrm{u}} = e_{\mathrm{b}}(T_{\mathrm{u}})$ and $J_{\mathrm{d}} = e_{\mathrm{b}}(T_{\mathrm{d}})$ when radiation is defined on both sides.
- When **Wavelength dependence of emissivity** is set to **Solar and ambient** or **Multiple spectral bands**, it defines for each spectral band $J_{\mathrm{B}i} = FEP_{\mathrm{B}i}(T)e_{\mathrm{b}}(T)$ when radiation is defined on one side or $J_{\mathrm{B}i,\,\mathrm{d}} = FEP_{\mathrm{B}i,\,\mathrm{d}}(T_{\mathrm{d}})e_{\mathrm{b}}(T_{\mathrm{d}})$ and $J_{\mathrm{u}} = FEP_{\mathrm{B}i,\,\mathrm{u}}(T_{\mathrm{u}})e_{\mathrm{b}}(T_{\mathrm{u}})$ when radiation is defined on both sides.

- When the temperature varies across a pair (for example when a Thin Layer condition is active on the same boundary), the temperature used to define the radiosity is evaluated on the side where the surface radiation is defined.
- The blackbody hemispherical total emissive power is defined by $e_{\mathrm{b}}(T) = n^2\sigma T^4$

Graybody Radiation

When **Graybody radiation** is selected it sets the surface radiosity expression corresponding to a graybody.

By default, the **Surface emissivity** ε is defined **From material**. In this case, make sure that a material is defined at the boundary level (materials are defined by default at the domain level). If **User defined** is selected for the **Surface emissivity**, enter another value for ε.

If **Wavelength dependence of emissivity** is set to **Constant**:

- When radiation is defined on one side, define the **Surface emissivity** ε to set $J = \varepsilon e_b(T)$, or
- When radiation is defined on both sides, define the **Material on upside**, the **Surface emissivity** ε_u, **Material on downside** and the **Surface emissivity** ε_d on the upside and downside, respectively. The surface radiosity on upside and downside is then defined by $J_u = \varepsilon_u e_b(T_u)$ and $J_d = \varepsilon_d e_b(T_d)$ respectively.

If **Wavelength dependence of emissivity** is set to **Solar and ambient** or **Multiple spectral bands**, for all spectral bands:

- When radiation is defined on one side for Bi spectral band, define the **Surface emissivity** ε_{Bi} to set $J_{Bi} = FEP_{Bi}\varepsilon_{Bi}e_b(T)$, or
- When radiation is defined on both sides for Bi spectral band, define the **Material on upside**, the **Surface emissivity** $\varepsilon_{Bi,\,u}$, **Material on downside** and the **Surface emissivity** $\varepsilon_{Bi,\,d}$ on the upside and downside, respectively. The surface radiosity on upside and downside is then defined by $J_u = FEP_{Bi}(T_u)\varepsilon_{Bi,\,u}e_b(T_u)$ and $J_d = FEP_{Bi}(T_d)\varepsilon_{Bi,\,d}e_b(T_d)$, respectively.

Set the surface emissivity to a number between 0 and 1, where 0 represents diffuse mirror and 1 is appropriate for a perfect blackbody. The proper value for a physical material lies somewhere in-between and can be found from tables or measurements.

User Defined

If **Wavelength dependence of emissivity** is set to **Constant** and **Radiosity expression** is set to **User defined**, it sets the surface radiosity expression to $J = J_0$, which specifies how the radiosity of a boundary is evaluated when that boundary is visible in the calculation of the irradiation onto another boundary in the model. Enter a **Surface radiosity expression**, J_0.

When the **Radiation direction** is set to **Both sides** (under Radiation Settings) also define the surface **Radiosity expression** $J_{0i,\,u}$ and $J_{0i,\,d}$ on the upside and downside, respectively. The geometric normal points from the downside to the upside.

If **Wavelength dependence of emissivity** is **Solar and ambient** or **Multiple spectral bands**, similar settings are available for each spectral band.

	Several settings for this node depend on the **Wavelength dependence of emissivity** setting, which is defined for the physics interface. See Radiation Settings.

	Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently. See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

	To define temperature dependencies for the user inputs (surface emissivity for example), use the temperature variable `ht.T`, that corresponds to the appropriate variable (upside, downside, or average temperature of a layer, wall temperature with turbulence modeling), depending on the model configurations. See Boundary Wall Temperature for a thorough description of the boundary temperature variables.

	Theory for Surface-to-Surface Radiation

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Surface-to-Surface Radiation>Radiation>Prescribed Radiosity
Surface-to-Surface Radiation>Prescribed Radiosity

More locations are available if the **Surface-to-surface radiation** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Radiation>Prescribed Radiosity

Ribbon

Physics Tab with **Surface-to-Surface Radiation** selected:

Boundaries>Surface-to-Surface Radiation>Prescribed Radiosity

Physics Tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>Radiation>Prescribed Radiosity

Radiation Group

This node enables you to specify radiation groups to speed up the radiation calculations and gather boundaries in a radiation problem that can see one another.

By default, all radiative boundaries (selected in a Diffuse Surface, Diffuse Mirror, or Prescribed Radiosity node) belong to the same radiation group.

To change this, select the **Use radiation groups** check box under **Radiation Settings** to add a **Radiation Group** to a Surface-to-Surface Radiation (rad) interface or any version of a Heat Transfer (ht) interface with the **Surface-to-surface radiation** check box selected.

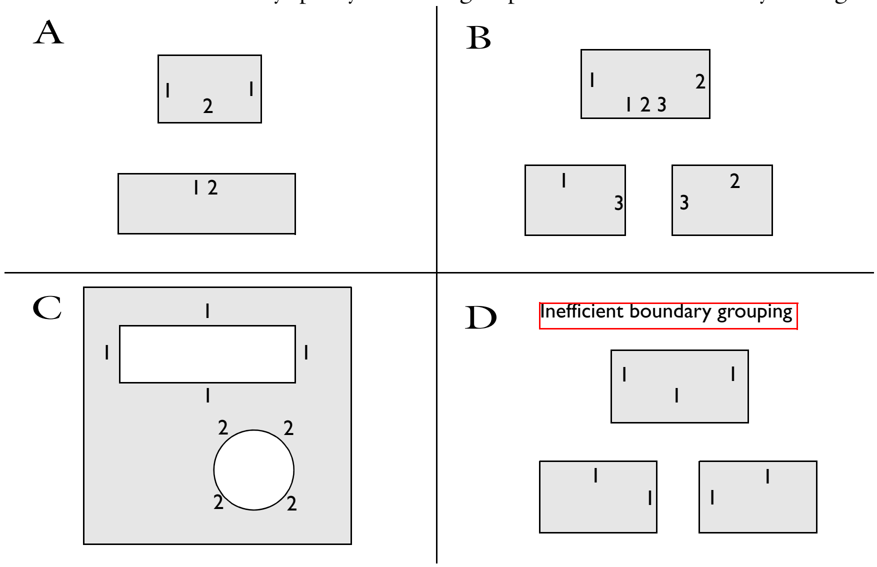
When a node is added to another radiation group, it is overridden in the default group. Then this boundary can be added to other radiation groups without being overridden by the manually added radiation groups.

Be careful when grouping boundaries in axisymmetric geometries. The grouping cannot be based on which boundaries have a free view toward each other in the 2D geometry. Instead, consider the full 3D geometry, obtained by revolving the model geometry about the z axis, when defining groups.

For example, parallel vertical boundaries must typically belong to the same group in 2D axisymmetric components, but to different groups in a planar model using the same 2D geometry.

The figure below shows four examples of possible boundary groupings. On boundaries that have no number, the user has NOT set a node among the Diffuse Surface, Diffuse Mirror, and Prescribed Radiosity nodes. These boundaries do not irradiate other boundaries, neither do other boundaries irradiate them.

On boundaries that belong to one or more radiation group, the user has set a node among the Diffuse Surface, Diffuse Mirror, and Prescribed Radiosity nodes. The numbers on each boundary specify different groups to which the boundary belongs.



Examples of radiation group boundaries.

To obtain optimal computational performance, it is good practice to specify as many groups as possible as opposed to specifying few but large groups. For example, case B is more efficient than case D.

BOUNDARY SELECTION

This section should contain any boundary that is selected in a **Diffuse Surface**, **Diffuse Mirror**, or **Prescribed Radiosity** node and that has a chance to see one of the boundary that is already selected in the **Radiation Group**.

RADIATION GROUP

When the **Wavelength dependence of emissivity** is **Constant**, the radiation group is valid for all wavelengths, and all this section is then empty.

When the **Wavelength dependence of emissivity** is set to **Solar and ambient** or **Multiple spectral bands**, the radiation group is defined for all spectral bands by default. Clear

Radiation group defined on spectral band i check boxes to remove the B_i spectral bands from these radiation groups.

Several settings for this node depend on the **Wavelength dependence of emissivity** setting, which is defined for the physics interface. See Radiation Settings.

If this node is combined with heat transfer in 2D and 1D, the thickness is assumed to be infinite for the view factor computation. The user-defined value for d is still used in the heat transfer equation.

Theory for Surface-to-Surface Radiation

LOCATION IN USER INTERFACE

Context menus

If the **Use radiation groups** check box is selected under the **Radiation Settings** section:

Heat Transfer with Surface-to-Surface Radiation>Radiation>Radiation Group
Surface-to-Surface Radiation>Radiation Group

More locations are available if the **Surface-to-surface radiation** check box is selected under the **Physical Model** section and the **Use radiation groups** check box is selected under the **Radiation Settings** section. For example:
Heat Transfer in Solids>Radiation>Radiation Group

Ribbon

Physics Tab with **Surface-to-Surface Radiation** selected:
Boundaries>Surface-to-Surface Radiation>Radiation Group

Physics Tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**,**Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:
Boundaries>Radiation>Radiation Group

Symmetry

This node provides a boundary condition for symmetry boundaries. This boundary condition is similar to a Thermal Insulation condition, and it means that there is no heat flux across the boundary.

The symmetry condition only applies to the temperature field. It has no effect on the radiosity (surface-to-surface radiation) and on the radiative intensity (radiation in participating media). Use Symmetry for Surface-to-Surface Radiation to account for the symmetry in the computation of the view factors.

Heat Generation in a Disc Brake: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Symmetry
Heat Transfer in Solids>Symmetry
Heat Transfer in Fluids>Symmetry
Heat Transfer in Porous Media>Symmetry
Bioheat Transfer>Symmetry
Heat Transfer with Surface-to-Surface Radiation>Symmetry
Heat Transfer with Radiation in Participating Media>Symmetry

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Symmetry**

Temperature

Use this node to specify the temperature somewhere in the geometry, for example, on boundaries.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

TEMPERATURE

The equation for this condition is $T = T_0$, where T_0 is the prescribed temperature on the boundary. For **User defined**, enter a value or expression for the **Temperature** T_0. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. By default **Classic constraints** is selected. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation. Select the **Discontinuous Galerkin constraints** button when **Classic constraints** do not work satisfactorily.

The **Discontinuous Galerkin constraints** option is especially useful to prevent oscillations on inlet boundaries where convection dominates. Unlike the **Classic constraints**, these constraints do not enforce the temperature on the boundary extremities. This is relevant on fluid inlets where the temperature condition should not be enforced on the walls at the inlet extremities. Note that **Discontinuous Galerkin contraints** are not supported for resistive thin layers or with turbulent wall functions.

Steady-State 2D Axisymmetric Heat Transfer with Conduction: Application Library path **Heat_Transfer_Module/Tutorials,_Conduction/cylinder_conduction**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Temperature
Heat Transfer in Solids>Temperature
Heat Transfer in Fluids>Temperature
Heat Transfer in Porous Media>Temperature
Bioheat Transfer>Temperature

Heat Transfer with Surface-to-Surface Radiation>Temperature
Heat Transfer with Radiation in Participating Media>Temperature

Heat Transfer in Solids>Pairs>Temperature
Heat Transfer in Fluids>Pairs>Temperature
Heat Transfer in Porous Media>Pairs>Temperature
Bioheat Transfer>Pairs>Temperature
Heat Transfer with Surface-to-Surface Radiation>Pairs>Temperature
Heat Transfer with Radiation in Participating Media>Pairs>Temperature

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Temperature**
Pairs>*interface***>Temperature**

Thermal Contact

This node defines correlations for the conductance h at the interface of two bodies in contact. It can be added to pairs by selecting **Pair Thermal Contact** from the **Pairs** menu. Note that in this case the source term is applied on the destination side.

The conductance h is involved in the heat flux across the surfaces in contact according to:

$$-\mathbf{n}_{\mathrm{d}} \cdot \mathbf{q}_{\mathrm{d}} = -h(T_{\mathrm{u}} - T_{\mathrm{d}}) + rQ_{\mathrm{b}}$$

$$-\mathbf{n}_{\mathrm{u}} \cdot \mathbf{q}_{\mathrm{u}} = -h(T_{\mathrm{d}} - T_{\mathrm{u}}) + (1-r)Q_{\mathrm{b}}$$

where u and d subscripts refer to the upside and downside of the slit, respectively. **Pair Thermal Contact** should be activated on a **Identity Pair** or on a **Contact Pair** where a structural mechanics physics interface defines a contact pair feature.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

THERMAL CONTACT

Constriction Conductance

Select a **Constriction conductance**: **Cooper-Mikic-Yovanovich correlation** (the default), **Mikic elastic correlation**, or **User defined**. For **User defined** enter a value or expression for h_{c}.

Gap Conductance

Select the **Gap conductance**: **User defined** (the default) or **Parallel-plate gap gas conductance** (available if **Cooper-Mikic-Yovanovich correlation** or **Mikic elastic correlation** is selected as the **Constriction conductance**). For **User defined** enter a value for h_{g}.

Radiative Conductance

When the **Surface-to-surface radiation** check box is selected under the **Physical Model** section on a physics interface, choose the **Radiative conductance**: **User defined** (the default) or **Gray-diffuse parallel surfaces**.

For **User defined** enter a value for h_{r}.

CONTACT SURFACE PROPERTIES

This section is available if **Cooper-Mikic-Yovanovich correlation** or **Mikic elastic correlation** are chosen as the **Constriction conductance correlation** under **Contact**. Enter values for the:

- **Surface roughness, asperities average height** σ_{asp}
- **Surface roughness, asperities average slope** m_{asp}
- **Contact pressure** p

For **Cooper-Mikic-Yovanovich correlation** select a **Hardness definition**: **Microhardness** (the default), **Vickers hardness**, or **Brinell hardness**.

- For **Microhardness** enter a value for H_{c}.
- For **Vickers hardness** enter a value for the **Vickers correlation coefficient** c_1 and **Vickers size index** c_2.
- For **Brinell hardness** enter a value for H_{B}. It should be between 1.30 and 7.60 GPa.

For **Mikic elastic correlation** select **Contact interface Young's modulus** E_{contact}: **Weighted harmonic mean** (the default) or **User defined**.

- For **Weighted harmonic mean** enter values or expressions for the **Young's modulus** E and **Poisson's ratio** ν.
- For **User defined** enter another value or expression for E_{contact}.

GAP PROPERTIES

This section is available when **Parallel-plate gap gas conductance** is selected as the **Gap conductance correlation** under **Thermal Contact**.

The default **Gas thermal conductivity** k_{gap} is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the gas thermal conductivity, and enter another value or expression.

Also enter the following:

- **Gas pressure** p_{gap}
- **Gas thermal accommodation parameter** α
- **Gas fluid parameter** β
- **Gas particles diameter** D

RADIATIVE CONDUCTANCE

This section is available when **Gray-diffuse parallel surfaces** is selected as the **Radiative conductance correlation** under **Thermal Contact**.

By default the **Surface emissivity** ε is taken **From material**. For **User defined** enter another value or expression.

THERMAL FRICTION

Select a **Heat partition coefficient** r: **Charron's relation** (the default) or **User defined**. For **User defined** enter a value for r.

Select either the **General source** (the default) or **Heat rate**.

- For **General source** enter a frictional heat source Q_{b}.
- For **Heat rate** enter the heat rate P_{b}.

Theory for Thermal Contact

Upside and downside settings can be visualized by plotting the global normal vector (`nx`, `ny`, `nz`), that always points from downside to upside. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.

See Tangent and Normal Variables in the *COMSOL Multiphysics Reference Manual*.

Thermal Contact Resistance Between an Electronic Package and a Heat Sink: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/thermal_contact_electronic_package_heat_sink**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Thermal Contact
Heat Transfer in Solids>Thermal Contact
Heat Transfer in Fluids>Thermal Contact
Heat Transfer in Porous Media>Thermal Contact
Bioheat Transfer>Thermal Contact
Heat Transfer with Surface-to-Surface Radiation>Thermal Contact
Heat Transfer with Radiation in Participating Media>Thermal Contact

Heat Transfer in Solids>Pairs>Pair Thermal Contact
Heat Transfer in Fluids>Pairs>Pair Thermal Contact
Heat Transfer in Porous Media>Pairs>Pair Thermal Contact
Bioheat Transfer>Pairs>Pair Thermal Contact
Heat Transfer with Surface-to-Surface Radiation>Pairs>Pair Thermal Contact
Heat Transfer with Radiation in Participating Media>Pairs>Pair Thermal Contact

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Pairs>*interface***>Pair Thermal Contact**

Thermal Insulation

This node is the default boundary condition for all Heat Transfer interfaces. This boundary condition means that there is no heat flux across the boundary:

$$-\mathbf{n} \cdot \mathbf{q} = 0$$

and hence specifies where the domain is well insulated. Intuitively, this equation says that the temperature gradient across the boundary is zero. For this to be true, the temperature on one side of the boundary must equal the temperature on the other side. Because there is no temperature difference across the boundary, heat cannot transfer across it.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Thermal Insulation
Heat Transfer in Solids>Thermal Insulation
Heat Transfer in Fluids>Thermal Insulation
Heat Transfer in Porous Media>Thermal Insulation
Bioheat Transfer>Thermal Insulation
Heat Transfer with Surface-to-Surface Radiation>Thermal Insulation
Heat Transfer with Radiation in Participating Media>Thermal Insulation

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:
Boundaries>*interface***>Thermal Insulation**

Thin Conductive Layer (Heat Transfer in Thin Shells Interface)

This node adds the heat equation for conductive heat transfer in shells (see Equation 4-45).

HEAT CONDUCTION, SOLID

By default the **Thermal conductivity** k uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** based on the characteristics of the thermal conductivity and enter other values or expressions in the field or matrix.

THERMODYNAMICS, SOLID

Specify the **Density** ρ and the **Heat capacity at constant pressure** C_p to describe the amount of heat energy required to produce a unit temperature change in a unit mass. The default settings use values **From material** for both. For **User defined** enter other values or expressions.

	Theory for Heat Transfer in Thin Structures Theory for Surface-to-Surface Radiation

	Disk-Stack Heat Sink: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/disk_stack_heat_sink**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Thin Conductive Layer

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

Boundaries>Heat Transfer in Thin Shells>Thin Conductive Layer

Thin Film

This node behaves like Fluid but is applicable on boundaries.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

THIN FILM

The available options for **Thin film model** are **Thermally thin approximation** and **General**. The former is a lumped model that accounts only for tangential temperature gradients (along the film), whereas the latter uses the Extra Dimension tool to account also for the normal gradients of temperature (through the film's thickness). The second model may be used for the modeling of bearings for example.

FILM PROPERTIES

This section is only available when the node is added from one of the versions of the Heat Transfer interface (`ht`). Enter a **Film thickness** d_f.

When the node is added from one of the versions of the Heat Transfer in Thin Shells interface (`htsh`), the shell's thickness is used instead.

LAYER DISCRETIZATION

This section is only available when the **Thin film model** is set to **General**.

Define the **Number of elements** used for the discretization of the film's thickness

> The settings for the **Model Inputs**, **Heat Conduction, Fluid**, and **Thermodynamics, Fluid** sections are the same as for Fluid.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Thin Film
Heat Transfer in Solids>Thin Film
Heat Transfer in Fluids>Thin Film
Heat Transfer in Porous Media>Thin Film
Bioheat Transfer>Thin Film
Heat Transfer with Surface-to-Surface Radiation>Thin Film
Heat Transfer with Radiation in Participating Media>Thin Film
Heat Transfer in Thin Shells>Thin Film
Heat Transfer in Thin Films>Thin Film
Heat Transfer in Fractures>Thin Film

Heat Transfer>Pairs>Thin Film
Heat Transfer in Solids>Pairs>Thin Film
Heat Transfer in Fluids>Pairs>Thin Film
Heat Transfer in Porous Media>Pairs>Thin Film
Bioheat Transfer>Pairs>Thin Film
Heat Transfer with Surface-to-Surface Radiation>Pairs>Thin Film
Heat Transfer with Radiation in Participating Media>Pairs>Thin Film

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Thin Film**
Pairs>*interface***>Thin Film**

Physics Tab with *interface* as **Heat Transfer in Thin Shells**, **Heat Transfer in Thin Films** or **Heat Transfer in Fractures** selected:
Boundaries>*interface***>Thin Film**

Thin Layer

This node defines the thickness and thermal conductivity of a material located on internal or external boundaries. This material can be formed of one or more layers, and different conductive behaviors can be modeled through the setting of the **Layer type**:

- Select **Thermally thick approximation** to model a layer that is a bad thermal conductor compared to the adjacent geometry. In this case, the tangential heat flux is neglected and only the heat flux across the layer's thickness is considered. The layer can be constituted of multiple sub-layers with specific thickness and conductivity. The up and down sides can be distinguished when a heat source is applied to the layer. This option may also be used to enforce consistent initial conditions.
- Select **Thermally thin approximation** to model a layer that is a good thermal conductor compared to the adjacent geometry. In this case, the temperature difference and heat flux across the layer's thickness are neglected. Only the tangential heat flux is considered. The up and down sides are not distinguished when a heat source is applied to the layer.
- Select **General** to model a layer in which both the normal and tangential heat fluxes should be considered. The layer can be constituted of multiple sub-layers with specific thickness and conductivity, and heat sources can be applied on a sub-layer selection, and on up and down sides of the layer.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

THIN LAYER

The available options for **Layer type** are **Thermally thick approximation**, **Thermally thin approximation**, and **General**.

Thermally thick approximation

From the **Specify** list select **Layer properties** (the default) or **Thermal resistance**.

- For **Layer properties** enter a value or expression for the **Layer thickness** d_s.
- For **Thermal resistance** enter a value or expression for the **Thermal resistance** R_s.
- By default the **Multiple layers** check box is not selected. For **Layer properties** click to select the check box to define multiple sandwiched thin layers with different thermal conductivities.

Thermally thin approximation

For **Thermally thin approximation** enter a value or expression for the **Layer thickness** d_s.

General

For **General** no additional setting is required in this section.

HEAT CONDUCTION

This section is not available when **Layer type** is set to **Thermally thick approximation** and **Specify** is set to **Thermal Resistance** in the **Thin Layer** settings.

Thermally thick approximation (Layer Type)

The default **Layer thermal conductivity** k_s is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** to enter another value or expression.

Thermally thin approximation (Layer Type)

For **Thermally thin approximation** the default **Layer thermal conductivity** k_s is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** to enter another value or expression. If the thickness is zero, the thin layer does not take effect.

General (Layer Type)

Select the **Number of layers** to define (1 to 5) and set the properties for each layer selected.

- Select an option from the **Layer** (**1**, **2**, **3**, **4**, or **5**) list to assign a material to each layer. The default **Boundary material** takes the material from the boundary.
- For each layer, enter the **Layer thickness** d_s.
- The default **Thermal conductivity** k_s is taken **From material**, which is then taken from the material selected in **Layer (1**, **2**, ...**)**. For **User defined** enter another value or expression.

THERMODYNAMICS

This section is available when the **Layer type** is **General** or **Thermally thin approximation**. Set the following properties for each **Layer**:

By default the **Layer density** ρ_s and **Layer heat capacity** $C_{p,\,s}$ values are taken **From material**. For **User defined** enter other values or expressions.

THIN LAYER OPACITY

This section is only available when the **Surface-to-surface radiation** check box is selected under the **Physical Model** section on the parent physics interface.

Select **Opaque** (the default) or **Transparent** to set the layer's opacity type.

This is needed when the thin layer (with **Layer type** as **Thermally thick approximation** or **General**) contributes with any boundary condition from the **Radiation** menu. It picks the side where irradiation starts from.

LAYER DISCRETIZATION

This section is only available when the **Layer type** is set to **General**.

Define the **Number of elements** used for discretizing the layer thickness.

These subnodes are available for **Thin Layer** node:

- Layer Heat Source (Thin Layer) — to add a layer internal heat source, Q_s, within the layer.
- Line Heat Flux (Thin Layer, Thin Film, Fracture) — to add a heat flux through a specified set of boundaries.
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells) — to set a prescribed temperature condition on a specified set of boundaries.
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture) — to add a surface-to-ambient radiation for the layer end.

When multiple layers are defined they are numbered from the upside (Layer 1) to the downside.

	Theory for Heat Transfer in Thin Structures Boundary Wall Temperature Plotting Results in Thin Layers Extra Dimensions

	• *Heat Transfer in a Surface-Mount Package for a Silicon Chip*: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/surface_mount_package** • *Silica Glass Block Coated with a Copper Layer*: Application Library path **Heat_Transfer_Module/Tutorials,_Thin_Structure/copper_layer**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Thin Layer
Heat Transfer in Solids>Thin Layer
Heat Transfer in Fluids>Thin Layer
Heat Transfer in Porous Media>Thin Layer
Bioheat Transfer>Thin Layer
Heat Transfer with Surface-to-Surface Radiation>Thin Layer
Heat Transfer with Radiation in Participating Media>Thin Layer

Heat Transfer>Pairs>Thin Layer
Heat Transfer in Solids>Pairs>Thin Layer
Heat Transfer in Fluids>Pairs>Thin Layer
Heat Transfer in Porous Media>Pairs>Thin Layer
Bioheat Transfer>Pairs>Thin Layer
Heat Transfer with Surface-to-Surface Radiation>Pairs>Thin Layer
Heat Transfer with Radiation in Participating Media>Pairs>Thin Layer

Ribbon

Physics Tab with *interface* as **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Boundaries>*interface***>Thin Layer**
Pairs>*interface***>Thin Layer**

Thin Layered Shell (Heat Transfer in Thin Shells Interface)

Use this node on the boundary of a thin shell. This overrides Thin Conductive Layer (Heat Transfer in Thin Shells Interface) to enable modeling multiple sandwich layers in the shell interface.

HEAT CONDUCTION

Select the **Number of layers** to define (1 to 5) and set the properties for each layer.

- Select an option from the **Layer** (**1**, **2**, **3**, **4**, or **5**) list to assign a material to each layer. The default setting, **Boundary material**, takes the material from the boundary.
- For each layer, enter the **Layer thickness** d_s.
- The default **Thermal conductivity** k_s is taken **From material**, which is then taken from the material selected in **Layer (1**, **2**, ...**)**. For **User defined** enter another value or expression.

THERMODYNAMICS

Set the following properties for each **Layer**.

By default the **Layer density** ρ_s and **Layer heat capacity** $C_{p,\,s}$ values are taken **From material**. For **User defined** enter other values or expressions.

LAYER DISCRETIZATION

This section is available when the **Layer type** is set to **General**. Define the **Number of elements per layer** used for the discretization of the thickness for each layer (the default is 2).

When multiple layers are defined they are numbered from the upside (Layer 1) to the downside.

Theory for Heat Transfer in Thin Structures

Plotting Results in Thin Layers Extra Dimensions

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Thin Layered Shell

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

Boundaries>Heat Transfer in Thin Shells>Thin Layered Shell

Edge Features

The Heat Transfer interfaces have the following edge nodes and subnodes available:

- Change Effective Thickness (Heat Transfer in Thin Shells Interface)
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Line Heat Source (Thin Rod)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Thin Rod

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Change Effective Thickness (Heat Transfer in Thin Shells Interface)

This node models edges (3D components) or point (2D and 2D axisymmetric components) with another thickness than the overall thickness that is specified in the Heat Transfer in Thin Shells interface **Shell Thickness** section (0.01 m by default). It defines the height of the part of the edge that is exposed to the ambient surroundings.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

CHANGE EFFECTIVE THICKNESS

Enter a value for the **Effective thickness** d_s. This value replaces the overall thickness in the selection.

Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Change Effective Thickness
Heat Transfer in Thin Shells>Pairs>Change Effective Thickness

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:
Support>**Heat Transfer in Thin Shells>Change Effective Thickness**
Pairs>Change Effective Thickness

with *Support* as **Egdes** in 3D and **Points** in 2D.

Heat Flux (Heat Transfer in Thin Shells Interface)

Use this node to add heat flux across boundaries of a thin shell. A positive heat flux adds heat to the domain. This feature adds a heat source (or sink) to edges. It adds a heat flux $q = d_s q_0$.

FRAME SELECTION

To display this section add both a **Heat Transfer in Thin Shells (htsh)** and **Moving Mesh (ale)** interface (found under the **Mathematics>Deformed Mesh** branch when adding a physics interface). Then click the **Show** button ( ) and select **Advanced Physics Options**. The rest of the settings are the same for the Heat Flux node as described under **Frame Selection**.

HEAT FLUX

These settings are the same as for the Heat Flux node available for the other interfaces.

In 2D, $q = d_s d_z q_0$ to account for the out-of-plane thickness.

Theory for Heat Transfer in Thin Structures

Handling Frames in Heat Transfer

Shell Conduction: Application Library path
Heat_Transfer_Module/Tutorials,_Thin_Structure/shell_conduction

LOCATION IN USER INTERFACE

Context menus
Heat Transfer in Thin Shells>Heat Flux

Ribbon
Physics Tab with **Heat Transfer in Thin Shells** selected:
Support>**Heat Transfer in Thin Shells>Heat Flux**

with *Support* as **Egdes** in 3D and **Points** in 2D.

Heat Source (Heat Transfer in Thin Shells Interface)

This node models a linear heat source (or sink). It adds a heat source $q = Q_1$, $q = d_s Q_b$, or $q = P_b / L$. A positive q is heating and a negative q is cooling.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

HEAT SOURCE

Click the **General source** (the default) or **Heat rate** button.

For **General source**, from the **Edge heat source type** list, select **Heat source defined per unit of length** (the default) or **Heat source defined per unit of area**.

- For **Heat source defined per unit of length** enter a value for the distributed heat source, Q_1 in unit power per unit length. Positive Q_1 is heating and a negative Q_1 is cooling.
- For **Heat source defined per unit of area** enter the boundary heat source Q_b. A positive Q_b is heating and a negative Q_b is cooling.

For **Heat rate** enter the heat rate P_b. In this case $Q_l = P_b / L$, where L is the total length of the selected edges.

FRAME SELECTION

The settings are the same for the Heat Source node **Frame Selection** section.

In 2D components, the equation contains an additional factor, d_z, to account for the out-of-plane thickness. This is because the selected points correspond to edges in a 3D geometry.

Theory for Heat Transfer in Thin Structures

Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Heat Source
Heat Transfer in Thin Shells>Pairs>Heat Source

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

*Support***>Heat Transfer in Thin Shells>Heat Source**

Pairs>Heat Source

with *Support* as **Egdes** in 3D and **Points** in 2D.

Insulation/Continuity (Heat Transfer in Thin Shells Interface)

This node is the default edge condition on thin shells. On external edges, this condition means that there is no heat flux across the edge:

$$-\mathbf{n} \cdot \mathbf{q} = 0$$

On internal edges, this condition means that the temperature field and its flux is continuous across the edge.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

Theory for Heat Transfer in Thin Structures

Shell Conduction: Application Library path **Heat_Transfer_Module/Tutorials,_Thin_Structure/shell_conduction**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Insulation/Continuity
Heat Transfer in Thin Shells>Pairs>Insulation/Continuity

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

Support>**Heat Transfer in Thin Shells>Insulation/Continuity**

Pairs>Insulation/Continuity

with *Support* as **Egdes** in 3D and **Points** in 2D.

Line Heat Source (Thin Rod)

Use this subnode to add an internal heat source, Q_l, within the rod. Add one or more heat sources.

LINE HEAT SOURCE

Select the **General source** (the default) or **Heat rate** button to define Q_l.

- For **General source** enter a value or expression for Q_l as a heat source per volume.
- For **Heat rate** define the heat rate P_l. In this case $Q_l = P_l / V$.

Thin Rod

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Edges>Thin Rod>Line Heat Source

More locations are available. For example:
Heat Transfer in Fluids>Edges>Thin Rod>Line Heat Source

Ribbon

Physics Tab with **Thin Rod** selected in the model tree:
Attributes>Line Heat Source

Line Heat Flux (Thin Layer, Thin Film, Fracture)

Use this subnode to add heat flux across boundaries of a thin layer, a thin film or a fracture. A positive heat flux adds heat to the layer.

HEAT FLUX

Click the **General inward heat flux** (the default), **Inward heat flux**, or **Heat rate** (3D components only) button.

- If **General inward heat flux** is selected, it adds q_0 to the total flux across the selected edges. Enter a value for q_0 to represent a heat flux that enters the layer. For example, any electric heater is well represented by this condition and its geometry can be omitted.
- If **Inward heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{\mathrm{ext}} - T)$. Enter the **Heat transfer coefficient** h and the **External temperature** T_{ext}. This latter value depends on the geometry and the ambient flow conditions. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.
- *3D Components*: If **Heat rate** is selected, it adds q_0 in the form $q_0 = P_0/A$ where $A = Ld_{\mathrm{s}}$ (for **Thin Layer**), $A = Ld_{\mathrm{f}}$ (for **Thin Film**), or $A = Ld_{\mathrm{fr}}$ (for **Fracture**), and L is equal to the length of the edge selection. Enter the heat rate P_0.

FRAME SELECTION

This section is available with 3D components. The settings are the same for the Heat Source node and described under **Frame Selection**.

When **Line Heat Flux** is applied on a pair, the flux is only applied on the edge adjacent to the source boundary which is in contact with the destination boundary. Consider adding another pair with opposite source and destination boundaries to apply a flux on the edge adjacent to the destination boundary and in contact with the source boundary.

See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

Thin Layer

Thin Film

Fracture

Theory for Heat Transfer in Thin Structures

Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Layer>Line Heat Flux
Heat Transfer in Solids>Thin Film>Line Heat Flux
Heat Transfer in Porous Media>Fracture>Line Heat Flux

Ribbon

Physics Tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:
Attributes>Line Heat Flux

Line Heat Source

This node models a heat source (or sink) that is so thin that it has no thickness in the model geometry. It is available in 3D on edges. In 2D and 2D axisymmetric, it is available on points.

In theory, the temperature in a line source in 3D is plus or minus infinity (to compensate for the fact that the heat source does not have any volume). The finite element discretization used in COMSOL Multiphysics returns a finite temperature distribution along the line, but that distribution must be interpreted in a weak sense.

LINE HEAT SOURCE

Click the **General source** (the default) or **Heat rate** button.

- If **General source** is selected, enter a value for the distributed heat source, Q_l in unit power per unit length. A positive Q_l corresponds to heating while a negative Q_l corresponds to cooling.
- If **Heat rate** is selected, enter the heat rate P_l.

HEAT SOURCE RADIUS

With the Heat Transfer Module, you can model the heat source explicitly and apply it on a cylinder around the line.

Select the **Specify heat source radius** check box to define the **Heat source radius** R. This averages the source on a cylinder of given radius around the line. This option avoids obtaining an increasing temperature value at the line when meshing finer than this radius. It makes use of the `diskavg` operator for averaging around the source.

FRAME SELECTION

The settings are the same as for the Heat Source node and are described under **Frame Selection**.

See Built-In Operators in the *COMSOL Multiphysics Reference Manual* for additional information about the `diskavg` operator.

- Handling Frames in Heat Transfer
- About the Heat Transfer Interfaces

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>*support***>Line Heat Source**
Heat Transfer in Solids>*support***>Line Heat Source**
Heat Transfer in Fluids>*support***>Line Heat Source**
Heat Transfer in Porous Media>*support***>Line Heat Source**
Bioheat Transfer>*support***>Line Heat Source**

Heat Transfer with Surface-to-Surface Radiation>*support***>Line Heat Source**
Heat Transfer with Radiation in Participating Media>*support***>Line Heat Source**

Ribbon

Physics Tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

*Support***>Line Heat Source**

with *Support* as **Egdes** in 3D and **Points** in 2D.

Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)

Use this node to add surface-to-ambient radiation to edges of a thin shell. The net inward heat flux from surface-to-ambient radiation is

$$-\mathbf{n}\cdot\mathbf{q} = d_{\mathrm{s}}\varepsilon\sigma(T_{\mathrm{amb}}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

SURFACE-TO-AMBIENT RADIATION

Surface emissivity

The default **Surface emissivity** ε (a dimensionless number between 0 and 1) is taken **From material**. For **User defined**, it should be specified. An emissivity of 0 means that the surface emits no radiation at all while an emissivity of 1 means that it is a perfect blackbody.

Ambient temperature

For **User defined**, enter an **Ambient temperature** T_{amb}. The default value is approximately room temperature, 293.15 K (20 °C). Else, select an **Ambient**

temperature defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

In 2D, the equation has an additional factor, d_z, to account for the out-of-plane thickness.

Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Surface-to-Ambient Radiation
Heat Transfer in Thin Shells>Pairs>Surface-to-Ambient Radiation

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:
Support>**Heat Transfer in Thin Shells>Surface-to-Ambient Radiation**
Pairs>Surface-to-Ambient Radiation

with *Support* as **Egdes** in 3D and **Points** in 2D.

Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)

Use this subnode to add surface-to-ambient radiation to lines (geometrical edges in 3D or geometrical points in 2D and 2D axisymmetric) that represent thin boundaries of a thin layer, a thin film, or a fracture.

The net inward heat flux from surface-to-ambient radiation is

$$\lim_{\partial S \to 0} \int_{\partial S} Q ds = d_s \varepsilon \sigma (T_{\mathrm{amb}}^4 - T^4)$$

where d_s is the layer thickness (replaced by d_f for a thin film, and by d_fr for a fracture), ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_amb is the ambient temperature.

SURFACE-TO-AMBIENT RADIATION

Ambient temperature

For **User defined**, enter an **Ambient temperature** T_{amb}. The default value is approximately room temperature, 293.15 K (20 °C). Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

Surface emissivity

The default **Surface emissivity** ε (a dimensionless number between 0 and 1) is taken **From material**. For **User defined**, it should be specified. An emissivity of 0 means that the surface emits no radiation at all while an emissivity of 1 means that it is a perfect blackbody.

Thin Layer

Thin Film

Fracture

Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Layer>Surface-to-Ambient Radiation
Heat Transfer in Solids>Thin Film>Surface-to-Ambient Radiation
Heat Transfer in Porous Media>Fracture>Surface-to-Ambient Radiation

Ribbon

Physics Tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:
Attributes>Surface-to-Ambient Radiation

Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)

Use this subnode to specify the temperature on a set of lines (geometrical edges in 3D or geometrical points in 2D and 2D axisymmetric) that represent the boundaries of a thin domain (layer, film, fracture, or shell). Only edges (3D) or points (2D and 2D axisymmetric) adjacent to the boundaries can be selected in the parent node.

TEMPERATURE

For **User defined**, enter a value or expression for the **Temperature** T_0. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

The equation for this condition is $T = T_0$ where T_0 is the prescribed temperature.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

Thin Layer

Thin Film

Fracture

Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Layer>Temperature
Heat Transfer in Solids>Thin Film>Temperature
Heat Transfer in Porous Media>Fracture>Temperature
Heat Transfer in Thin Shells>Temperature
Heat Transfer in Thin Shells>Pairs>Temperature

Ribbon

Physics Tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:
Attributes>Temperature

Physics Tab with **Heat Transfer in Thin Shells** selected in the model tree:
*Support***>Heat Transfer in Thin Shells>Temperature**
Pairs>Temperature
with *Support* as **Egdes** in 3D and **Points** in 2D.

Shell Conduction: Application Library path
Heat_Transfer_Module/Tutorials,_Thin_Structure/shell_conduction

Thin Rod

Use this node to define the thermal and radius properties of conductive rods located on edges in a 3D component.

THIN ROD

The **Rod radius** r_1 should be specified.

HEAT CONDUCTION

The **Thermal conductivity** k_1 should be specified. By default it is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Anisotropic** to enter another value or expression.

THERMODYNAMICS

By default the **Density** ρ_1 and the **Heat capacity at constant pressure** $C_{p,1}$ values are taken **From material**. For **User defined** enter other values or expressions.

These additional subnodes are available for the **Thin Rod** node:

- Line Heat Source (Thin Rod) — to add an internal heat source, Q_1, within the rod.
- Temperature (Thin Rod) — to set a prescribed temperature condition on a specified set of points.
- Point Heat Flux (Thin Rod) — to add a heat flux through a specified set of points.
- Surface-to-Ambient Radiation (Thin Rod) — to add surface-to-ambient radiation at the rod end points.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer>Edges>Thin Rod
Heat Transfer in Solids>Edges>Thin Rod
Heat Transfer in Fluids>Edges>Thin Rod
Heat Transfer in Porous Media>Edges>Thin Rod
Bioheat Transfer>Edges>Thin Rod
Heat Transfer with Surface-to-Surface Radiation>Edges>Thin Rod
Heat Transfer with Radiation in Participating Media>Edges>Thin Rod

Ribbon

Physics Tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Edges>Thin Rod

Point Features

The Heat Transfer interfaces have the following point nodes and subnodes available:

- Change Effective Thickness (Heat Transfer in Thin Shells Interface)
- Heat Flux (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Heat Source (Heat Transfer in Thin Shells Interface)
- Insulation/Continuity (Heat Transfer in Thin Shells Interface)
- Line Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Heat Transfer in Thin Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture)
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Thin Shells)
- Temperature (Thin Rod)

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Heat Source (Heat Transfer in Thin Shells Interface)

This node, available in 3D and 2D axisymmetric components only, models a point heat source (or sink).

POINT HEAT SOURCE

Enter the **Point heat source** Q_p in unit power. A positive Q_p is heating and a negative Q_p is cooling.

HEAT SOURCE RADIUS

Select the **Specify heat source radius** check box to define the **Heat source radius** R. This setting averages the source on a cylinder of a given radius around the line, and avoids getting an increasing temperature value at the line when meshing finer than this radius. It makes use of the `diskavg` operator for averaging around the source.

FRAME SELECTION

This section is not available if the **Specify heat source radius option** is disabled. The settings are the same for the Heat Source node **Frame Selection** section

See Built-In Operators in the *COMSOL Multiphysics Reference Manual* for additional information about the `diskavg` operator.

Theory for Heat Transfer in Thin Structures

Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Thin Shells>Points>Heat Source

Ribbon

Physics Tab with **Heat Transfer in Thin Shells** selected:

Points>Heat Source

Point Heat Flux (Thin Rod)

Use this subnode to add heat flux at points of a thin rod. A positive heat flux adds heat to the rod.

HEAT FLUX

Select either the **General inward heat flux** (the default) or **Inward heat flux** buttons.

- If **General inward heat flux** is selected, it adds q_0 to the total flux across the selected points. Enter a value for q_0 to represent a heat flux that enters the rod.
- If **Inward heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{\mathrm{ext}} - T)$. Enter the **Heat transfer coefficient** h and the **External temperature** T_{ext}. The value depends on the geometry and the ambient flow conditions.

Thin Rod

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Rod>Point Heat Flux

More locations are available. For example:
Heat Transfer in Fluids>Thin Rod>Point Heat Flux

Ribbon

Physics Tab with **Thin Rod** selected in the model tree:
Attributes>Point Heat Flux

Point Heat Source

This node, available for 3D components, models a heat source (or sink) that is so small that it can be considered to have no spatial extension.

In theory, the temperature in a point source in 3D is plus infinity (to compensate for the fact that the heat source does not have a spatial extension). The finite element discretization used in COMSOL Multiphysics returns a finite value, but that value must be interpreted in a weak sense.

POINT HEAT SOURCE

Enter the **Point heat source** Q_{p} in unit power. A positive Q_{p} corresponds to heating while a negative Q_{p} corresponds to cooling.

HEAT SOURCE RADIUS

With the Heat Transfer Module, you can model the heat source explicitly and apply it on a ball or disk around the point.

Select the **Specify heat source radius** check box to define the **Heat source radius** R. This setting averages the source on a ball or disk of given radius around the point, and avoids obtaining an increasing temperature shift at the point when meshing finer than this radius. It makes use of the `ballavg` or `diskavg` operator for averaging around the source.

FRAME SELECTION

This section is not available if the **Specify heat source radius option** is disabled. The settings are the same as for the Heat Source node and are described under **Frame Selection**.

See Built-In Operators in the *COMSOL Multiphysics Reference Manual* for additional information about the `ballavg` and `diskavg` operators.

- Handling Frames in Heat Transfer
- About the Heat Transfer Interfaces

Heat Conduction with a Localized Heat Source on a Disk: Application Library path **Heat_Transfer_Module/Verification_Examples/localized_heat_source**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Points>Point Heat Source
Heat Transfer in Fluids>Points>Point Heat Source
Heat Transfer in Porous Media>Points>Point Heat Source
Bioheat Transfer>Points>Point Heat Source
Heat Transfer with Surface-to-Surface Radiation>Points>Point Heat Source
Heat Transfer with Radiation in Participating Media>Points>Point Heat Source

Ribbon

Physics Tab with **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with**

Surface-to-Surface Radiation or **Heat Transfer with Radiation in Participating Media** selected:

Points>Point Heat Source

Point Heat Source on Axis

This node, available for 2D axisymmetric components, models a heat source (or sink) that is so small that it can be considered to have no spatial extension.

The settings are the same as for the Point Heat Source node.

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Points>Point Heat Source on Axis
Heat Transfer in Fluids>Points>Point Heat Source on Axis
Heat Transfer in Porous Media>Points>Point Heat Source on Axis
Bioheat Transfer>Points>Point Heat Source on Axis
Heat Transfer with Surface-to-Surface Radiation>Points>Point Heat Source on Axis
Heat Transfer with Radiation in Participating Media>Points>Point Heat Source on Axis

Ribbon

Physics Tab with **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Points>Point Heat Source on Axis

Surface-to-Ambient Radiation (Thin Rod)

Use this subnode to add surface-to-ambient radiation to points that represent boundaries of a thin rod.

The net inward heat flux from surface-to-ambient radiation is

$$\lim_{\partial S \to 0} \int_{\partial S} Q ds = d_s \varepsilon \sigma (T_{amb}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

SURFACE-TO-AMBIENT RADIATION

Ambient temperature

For **User defined**, enter an **Ambient temperature** T_{amb}. The default value is approximately room temperature, 293.15 K (20 °C). Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

Surface emissivity

The default **Surface emissivity** ε (a dimensionless number between 0 and 1) is taken **From material**. For **User defined**, it should be specified. An emissivity of 0 means that the surface emits no radiation at all while an emissivity of 1 means that it is a perfect blackbody.

Thin Rod

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Rod>Surface-to-Ambient Radiation

More locations are available. For example:
Heat Transfer in Fluids>Thin Rod>Surface-to-Ambient Radiation

Ribbon

Physics Tab with **Thin Rod** selected in the model tree:
Attributes>Surface-to-Ambient Radiation

Temperature (Thin Rod)

Use this subnode to specify the temperature on a set of points that represent boundaries of a rod. Only points adjacent to the boundaries can be selected in the parent node.

TEMPERATURE

For **User defined**, enter a value or expression for the **Temperature** T_0. Else, select an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface. The equation for this condition is $T = T_0$ where T_0 is the prescribed temperature on the points.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**.

	Thin Rod

LOCATION IN USER INTERFACE

Context menus

Heat Transfer in Solids>Thin Rod>Temperature

More locations are available. For example:
Heat Transfer in Fluids>Thin Rod>Temperature

Ribbon

Physics Tab with **Thin Rod** selected in the model tree:

Attributes>Temperature

Global Features

The Heat Transfer interfaces have the following global nodes available:

- External Radiation Source
- Symmetry for Surface-to-Surface Radiation

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

External Radiation Source

Use this node in 2D and 3D components to define an external radiation source as a point or directional radiation source with view factor calculation. Each **External Radiation Source** node contributes to the incident radiative heat flux on all spectral bands, $G_{\mathrm{B}i}$ on all the boundaries where a **Diffuse Surface** or **Diffuse Mirror** boundary condition is active. The source contribution, $G_{\mathrm{ext,\,B}i}$, is equal to the product of the view factor of the source by the source radiosity. For radiation sources located on a point, $\mathrm{G}_{\mathrm{ext,\,B}i}=F_{\mathrm{ext,\,B}i}\,P_{\mathrm{s,\,B}i}$. For directional radiative source $G_{\mathrm{ext,\,B}i}=F_{\mathrm{ext,\,B}i}\,q_{0,\,\mathrm{s}}$.

- The external radiation sources are ignored on the boundaries when neither **Diffuse Surface** nor **Diffuse Mirror** is active.
- If this feature is combined with heat transfer in 2D and 1D, the thickness is assumed to be infinite for the view factor computation. The user-defined value for d is still used in the heat transfer equation.

SOURCE

Select a **Source position**: **Point coordinate** (the default) or **Infinite distance**. In 3D, **Solar position** is also available.

Point Coordinate

For **Point coordinate** define the **Source location** $\mathbf{x}_\mathrm{s}$ and the **Source power** P_s. The source radiates uniformly in all directions.

If **Wavelength dependence of emissivity** is **Solar and ambient** or **Multiple spectral bands**, set the **Source power definition** to **Blackbody** or **User defined**. When **Blackbody** is selected, enter the **Source temperature**, T_s, to define the source power on the spectral band B_i as $P_{s,\,\mathrm{B}i} = FEP_{\mathrm{B}i}(T_s)P_s$ where $FEP_{\mathrm{B}i}(T_s)$ is the fractional blackbody emissive power over B_i interval at T_s. When **User defined** is selected, enter an expression to define the source power on each spectral band B_i, $P_{s,\,\mathrm{B}i}$.

!	$\mathbf{x}_s$ should not belong to any surface where a **Diffuse Surface** or **Diffuse Mirror** boundary condition is active.

Infinite Distance

For **Infinite distance** define the **Incident radiation direction** $\mathbf{i}_s$ and the **Source heat flux** $q_{0,s}$.

If **Wavelength dependence of emissivity** is **Solar and ambient** or **Multiple spectral bands**, set the **Source heat flux definition** to **Blackbody** or **User defined**.

- When **Blackbody** is selected, enter the **Source temperature**, T_s, to define the source heat flux on the spectral band B_i as $q_{0,\,s,\,\mathrm{B}i} = q_{0,\,s}FEP_{\mathrm{B}i}(T_s)q_{0,\,s}$ where $FEP_{\mathrm{B}i}(T_s)$ is the fractional blackbody emissive power over B_i interval at T_s.
- When **User defined** is selected, enter an expression to define the source heat flux on each spectral band B_i, $q_{0,\,s,\,\mathrm{B}i}$.

Solar Position

	Solar position is available for 3D components. When this option is selected, use it to estimate the external radiative heat source due to the Sun. North, west, and the up directions correspond to the *x*, *y*, and *z* directions, respectively.

Select an option from the **Location defined by** list: **Coordinates** (the default) or **City**.

For **City** select a predefined city and country combination from the list. Click to select the **Include daylight saving time (Time zone + 1)** check box to add one hour to the default setting for the city selected. For example, if **New York City, USA** is selected and the default standard time zone is UTC–5 hours, when the check box is selected, the daylight saving time is used instead (UTC–4 hours).

If **Coordinates** is selected, or your city is not listed in the **Location defined by** table, define the following parameters:

- **Latitude**, a decimal value, positive in the northern hemisphere (the default is Greenwich UK latitude, 51.477). Enter a value without a unit to avoid double conversion. This is because the latitude value is expected to represent degrees but the model's unit for angles may be different (for example, the SI unit for the angle is radians).
- **Longitude**, a decimal value, positive at the east of the Prime Meridian (the default is Greenwich UK longitude, –0.0005). Enter a value without a unit to avoid double conversion. This is because the latitude value is expected to represent degrees but the model's unit for angles may be different (for example, the SI unit for the angle is radians).
- **Time zone**, the number of hours to add to UTC to get local time (the default is Greenwich UK time zone, 0). For example in **New York City, USA** the time zone is UTC–5 hours (standard time zone) or UTC–4 hours (with daylight saving time).

For either selection, in the **Date** table enter the:

- **Day**, the default is 01. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent days but the model's unit for time may be different (for example, the SI unit for time is seconds).
- **Month**, the default is 6 (June). Enter a value without a unit to avoid double conversion. This is because the value is expected to represent months but the model's unit for time may be different (for example, the SI unit for time is seconds).
- **Year**, the default is 2012. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent years but the model's unit for time may be different (for example, the SI unit for time is seconds). The solar position is accurate for a date between 2000 and 2199.

For either selection, in the **Local time** table enter the:

- **Hour**, the default is 12. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent hours but the model's unit for time may be different (for example, the SI unit for time is seconds).
- **Minute**, the default is 0. Enter a value without a unit to avoid double conversion. This is because the value is expected to represent minutes but the model's unit for time may be different (for example, the SI unit for time is seconds).
- **Second**, the default is 0.

For temporal studies, these inputs define the starting time of the simulation. By default, the **Update time from solver** check box is selected, and the time is then automatically updated with the time from the solver. Unselect this check box to manually set the time update.

For either selection, in the **Solar irradiance** field I_s define the incident radiative intensity coming from the sun. I_s represents the heat flux received from the sun by a surface perpendicular to the sun rays. When surfaces are not perpendicular to the sun rays the heat flux received from the sun depends on the incident angle.

For **User defined**, enter a value or expression for the **Solar irradiance** I_s. Else, select an **Ambient solar irradiance** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

If **Wavelength dependence of emissivity** is **Solar and ambient** or **Multiple spectral bands**, the solar irradiance is divided among all spectral bands B_i as $q_{0,\,s,\,Bi} = q_{0,s}FEP_{Bi}(T_{sun})q_{0,\,s}$ where $FEP_{Bi}(T_{sun})$ is the fractional blackbody emissive power over B_i interval at $T_{sun} = 5780$ K.

The **Wavelength dependence of emissivity** is defined in the physics interface settings, in the Radiation Settings section. When only one spectral band is defined, the Bi subscript in variable names is removed.

The sun position is updated if the location, date, or local time changes during a simulation. In particular for transient analysis, if the unit system for the time is in seconds (the default), the time change can be taken into account by adding `t` to the **Second** field in the **Local time** table. Note that no validity range is prescribed on the time inputs. It is possible to enter values that exceed the expected boundary. For example, entering 5h 2min 81s is equivalent to 5h 3min 21s. This makes it possible to enter `t` in the second field, even if the solution is computed for more than 60s.

The Heat Transfer with Surface-to-Surface Radiation Interface

Theory for Surface-to-Surface Radiation

	Sun's Radiation Effect on Two Coolers Placed Under a Parasol: Application Library path **Heat_Transfer_Module/Thermal_Radiation/parasol**

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Surface-to-Surface Radiation>Global>External Radiation Source
Surface-to-Surface Radiation>Global>External Radiation Source

More locations are available if the **Surface-to-surface radiation** check box is selected under the **Physical Model** section. For example:
Heat Transfer in Solids>Global>External Radiation Source

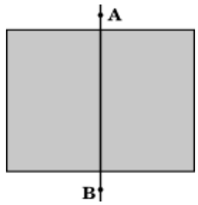
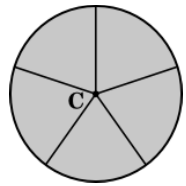
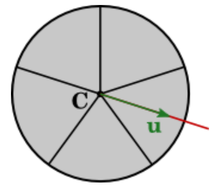
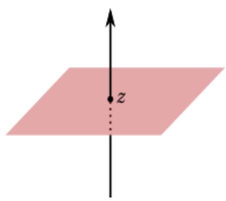
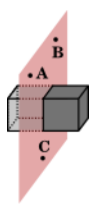
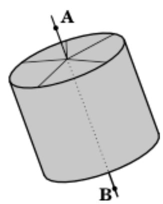
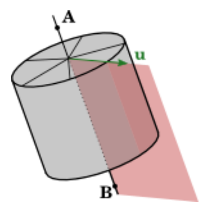
Ribbon

Physics Tab with **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with Surface-to-Surface Radiation**, **Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:
Global>External Radiation Source

Symmetry for Surface-to-Surface Radiation

Use this node to compute view factors on only a part of a symmetric geometry to improve efficiency, by defining either a symmetry plane in 2D, 2D axisymmetric, and 3D components; or sectors of symmetry in 2D and 3D components. In addition, a reflection plane can be defined inside each sector of symmetry. Table 6-4 summarizes the available options for each dimension.

TABLE 6-4: OPTIONS FOR SYMMETRY DEFINITION, EXAMPLE WITH 5 SECTORS

DIMENSION	PLANE	SECTORS	SECTORS AND REFLECTION
2D			
2D axi		Not applicable	Not applicable
3D			

SYMMETRY FOR SURFACE-TO-SURFACE RADIATION

Select the **Type of symmetry** to be defined between **Plane of symmetry** (the default) or **Sectors of symmetry**. Depending on the **Type of symmetry** selected from the list and on the dimension, further settings are required.

Plane of symmetry

If **Plane of symmetry** was selected, the coordinates of the points defining the plane need to be set:

- In 2D components, the symmetry plane is defined by two points. Set the **x** and **y** coordinates of the **First point on plane of reflection**, $\mathbf{A}$, and of the **Second point on plane of reflection**, $\mathbf{B}$.

- In 2D axisymmetric components, the symmetry plane is parallel to the z=0 plane. Set the **z coordinate of plane of reflection**.
- In 3D components, the symmetry plane is defined by three points. Set the **x**, **y**, and **z** coordinates of the **First point on plane of reflection**, $\mathbf{A}$, the **Second point on plane of reflection**, $\mathbf{B}$, and the **Third point on plane of reflection**, $\mathbf{C}$.

Sectors of symmetry

If **Sectors of symmetry** was selected, the coordinates of the points defining the symmetry axis need to be set:

- In 2D components, the symmetry axis is the out-of-plane vector, and the center of the symmetry must be defined. Set the **x** and **y** coordinates of the **Point of central symmetry**, $\mathbf{C}$.
- In 3D components, the symmetry axis is defined by two points. Set the **x**, **y**, and **z** coordinates of the **First point defining sector symmetry axis**, $\mathbf{A}$ and the **Second point defining sector symmetry axis**, $\mathbf{B}$.

Enter a value for the **Number of sectors**. This should be a numerical value greater or equal to 2.

If the **Reflection for symmetrical sector** check box is selected, set the coordinates of the **Radial direction of reflection plane**, $\mathbf{u}$. This option may be used when each sector has itself a plane of symmetry.

The Heat Transfer with Surface-to-Surface Radiation Interface

Theory for Surface-to-Surface Radiation

LOCATION IN USER INTERFACE

Context menus

Heat Transfer with Surface-to-Surface Radiation>Global>
Symmetry for Surface-to-Surface Radiation
Surface-to-Surface Radiation>Global>Symmetry for Surface-to-Surface Radiation

More locations are available if the **Surface-to-surface radiation** check box is selected under the **Physical Model** section. For example:

Heat Transfer in Solids>Global>Symmetry for Surface-to-Surface Radiation

Ribbon

Physics Tab with **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, **Bioheat Transfer**, **Heat Transfer with**

Surface-to-Surface Radiation, **Surface-to-Surface Radiation** or **Heat Transfer with Radiation in Participating Media** selected:

Global>Symmetry for Surface-to-Surface Radiation

7

The Moisture Transport Features

The Moisture Transport Interface has domain, boundary, and pair nodes available. These nodes, listed in alphabetical order in this section, are available from the **Physics** ribbon toolbar (Windows users), from the **Physics** context menu (Mac or Linux users), or by right-clicking to access the context menu (all users).

In this section:

Domain Features

The Moisture Transport interface has the following domain nodes available:

- Initial Values
- Moisture Source
- Porous Medium

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Initial Values

This node adds an initial value for the relative humidity that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. In addition to the default **Initial Values** node always present in the interface, you can add more **Initial Values** nodes if needed.

INITIAL VALUES

For **User defined**, enter a value or expression for the initial value of the **Relative humidity** ϕ. The default value is 0.5. Else, select an **Ambient relative humidity** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Initial Values

Ribbon

Physics Tab with **Moisture Transport** selected:

Domains>Moisture Transport>Initial Values

Moisture Source

This node describes moisture generation within the domain. You express addition and removal of moisture content with positive and negative values, respectively. Add one or more nodes as needed — all moisture sources within a domain contribute with each other.

The **Moisture Source** node adds a source term G to the right-hand side of the moisture transport equation:

$$G = G_0$$

MOISTURE SOURCE

Enter a value or expression for the **Moisture Source** G_0 per unit volume.

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Moisture Source

Ribbon

Physics Tab with **Moisture Transport** selected:

Domains>Moisture Transport>Moisture Source

Porous Medium

Use this node to model moisture transfer in a porous medium through vapor diffusion and capillary moisture flows. The moisture content variation is expressed through the transfer of relative humidity

$$\xi\frac{\partial\phi}{\partial t} + \nabla\cdot\mathbf{g} = G \tag{7-1}$$

$$\mathbf{g} = -(\xi D_{\mathrm{w}}\nabla\phi + \delta_{\mathrm{p}}\nabla(\phi p_{\mathrm{sat}})) \tag{7-2}$$

with the following material properties, fields, and source:

- ξ (SI unit: kg/m^3) is the moisture storage capacity.
- δ_{p} (SI unit: s) is the vapor permeability.
- ϕ (dimensionless) is the relative humidity.
- p_{sat} (SI unit: Pa) is the vapor saturation pressure.
- T (SI unit: K) is the temperature.
- D_{w} (SI unit: m^2/s) is the moisture diffusivity.
- G (SI unit: kg/(m^3·s)) is the moisture source (or sink). See Moisture Source node.

For a steady-state problem, the relative humidity does not change with time and the first term disappears.

MODEL INPUTS

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups are added, the model inputs appear here.

The default **Temperature** T and **Absolute pressure** p_A are **User defined**. When additional physics interfaces are added to the model, the temperature and absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Heat Transfer in Building Materials** interface is added, you can select **Temperature (ht)** from the list. If a **Laminar Flow** interface is added, you can select **Absolute pressure (spf)** from the list.

If the node was added automatically after selecting the **Heat and Moisture Transport** predefined multiphysics interface, the temperature of the **Heat and Moisture** multiphysics node is used by default and the input field is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** ( ).

POROUS MEDIUM

This section sets the material properties for moisture diffusivity, moisture storage, and vapor diffusion.

The default **Moisture diffusivity** D_w is taken **From material**. For **User defined**, set a value to characterize the liquid transport in function of the moisture content.

The default **Moisture storage function** w is taken **From material**. For **User defined**, set a value to characterize the relationship between the amount of accumulated water and the relative humidity in the material.

Two options are available for the specification of the material properties for vapor diffusion:

- **Vapor permeability** (default) to define the vapor permeability δ_p directly. The default is taken **From material**. For **User defined**, set a value.
- **Vapor resistance factor** μ to define the vapor permeability δ_p as:

$$\delta_p = \frac{\delta}{\mu}$$

where δ (SI unit: s) is the vapor permeability of still air. The default **Vapor resistance factor** is taken **From material**. For **User defined**, set a value.

The Moisture Transport Interface

The Heat and Moisture Transport Interface

Heat and Moisture

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Porous Medium

Ribbon

Physics Tab with **Moisture Transport** selected:

Domains>Moisture Transport>Porous Medium

Boundary Features

The Moisture Transport interface has the following boundary nodes available:

- Continuity
- Insulation
- Moisture Content
- Moisture Flux
- Symmetry
- Thin Moisture Barrier

For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries into account and automatically adds an **Axial Symmetry** node that is valid on the axial symmetry boundaries only.

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Continuity

This node can be added to pairs. It prescribes that the relative humidity is continuous across the pair. **Continuity** is only suitable for pairs where the boundaries match.

PAIR SELECTION

Choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Pairs>Continuity

Ribbon

Physics Tab with **Moisture Transport** selected:

Pairs>Moisture Transport>Continuity

Insulation

This node is the default boundary condition for the Moisture Transport interface. This boundary condition means that there is no moisture flux across the boundary:

$$-\mathbf{n} \cdot \mathbf{g} = 0$$

and hence specifies where the domain is insulated. Intuitively, this equation says that the relative humidity gradient across the boundary is zero. For this to be true, the relative humidity on one side of the boundary must equal the relative humidity on the other side. Because there is no relative humidity difference across the boundary, moisture cannot transfer across it. It can be applied on exterior boundaries only.

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Insulation

Ribbon

Physics Tab with **Moisture Transport** selected:

Boundaries>Moisture Transport>Thermal Insulation

Moisture Content

Use this node to specify the relative humidity on interior and exterior boundaries.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair on which to apply this condition. A pair has to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

MOISTURE CONTENT

The equation for this condition is $\phi = \phi_0$, where ϕ_0 is the prescribed relative humidity on the boundary. For **User defined**, enter a value or expression for the **Relative humidity** ϕ_0. The default value is 0.5. Else, select an **Ambient relative humidity** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Moisture Content

Moisture Transport>Pairs>Moisture Content

Ribbon

Physics Tab with **Moisture Transport** selected:

Boundaries>Moisture Transport>Moisture Content

Pairs>Moisture Transport>Moisture Content

Moisture Flux

Use this node to add moisture flux across exterior boundaries. A positive moisture flux adds moisture to the domain.

MOISTURE FLUX

Click to select the **General moisture flux** (the default) or **Convective moisture flux** button.

General Moisture Flux

It adds g_0 to the total flux across the selected boundaries. Enter a positive value for g_0 to represent a moisture flux that enters the domain.

Convective Moisture Flux

Enter a value for the **Moisture transfer coefficient** β. In addition, two options are available to specify the external conditions:

- If **Relative humidity** is selected (the default), set the **External relative humidity**, ϕ_{ext} and the **External temperature**, T_{ext} (used for the computation of the vapor saturation pressure). For **User defined**, enter values or expressions. Else, select an **Ambient relative humidity** and an **Ambient temperature** defined in the Ambient Settings section of a Heat Transfer or Heat Transfer in Shells interface. Convective moisture flux is defined by $g_0 = \beta(\phi_{\mathrm{ext}} p_{\mathrm{sat}}(T_{\mathrm{ext}}) - \phi p_{\mathrm{sat}}(T))$.
- If **Partial vapor pressure** is selected, enter an **External partial vapor pressure**, $p_{\mathrm{v,ext}}$. Convective moisture flux is defined by $g_0 = \beta(p_{\mathrm{v,ext}} - \phi p_{\mathrm{sat}}(T))$.

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Moisture Flux

Ribbon

Physics Tab with **Moisture Transport** selected:

Boundaries>Moisture Transport>Moisture Flux

Symmetry

This node provides a boundary condition for symmetry boundaries. This boundary condition is similar to an Insulation condition, and it means that there is no moisture flux across the boundary. It can be applied on exterior boundaries only.

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Symmetry

Ribbon

Physics Tab with **Moisture Transport** selected:

Boundaries>Moisture Transport>Symmetry

Thin Moisture Barrier

Use this node to model a discontinuous moisture content across interior boundaries. It adds a moisture flux proportional to the difference of relative humidity upside and downside the selected boundaries.

MODEL INPUTS

This section has fields and values that are inputs to expressions that define material properties for moisture diffusivity and moisture storage function. If such user-defined property groups are added, the model inputs appear here.

THIN MOISTURE BARRIER

Two options are available to define the moisture flux:

- If **Moisture transfer coefficient** is selected (the default), enter a value for the **Moisture transfer coefficient** β. The upside and downside moisture fluxes are defined by $\beta(\phi_d - \phi_u)$ and $\beta(\phi_u - \phi_d)$, respectively.
- If **Barrier material properties** is selected, you then specify the **Layer thickness** d_s, the **Moisture diffusivity** D_w, and the **Moisture content** w. The default **Moisture diffusivity** and **Moisture storage function** are taken **From material**. For **User defined**, enter values

or expressions into the text fields. The upside and downside moisture fluxes are defined by:

$$-\mathbf{n}\cdot\mathbf{g}_{\mathrm{u}} = \frac{\frac{\partial w}{\partial\phi}D_{\mathrm{w}}(\phi_{\mathrm{d}}-\phi_{\mathrm{u}})}{d_{\mathrm{s}}}$$

$$-\mathbf{n}\cdot\mathbf{g}_{\mathrm{d}} = \frac{\frac{\partial w}{\partial\phi}D_{\mathrm{w}}(\phi_{\mathrm{u}}-\phi_{\mathrm{d}})}{d_{\mathrm{s}}}$$

LOCATION IN USER INTERFACE

Context menus

Moisture Transport>Thin Moisture Barrier

Ribbon

Physics Tab with **Moisture Transport** selected:

Boundaries>Moisture Transport>Thin Moisture Barrier

8

Multiphysics Interfaces

The Heat Transfer Module includes predefined multiphysics interfaces for conjugate heat transfer, thermoelectric effect, and local thermal non-equilibrium modeling.

All these interfaces except the Local Thermal Non-Equilibrium interface and the Heat and Moisture Transport interface couple an interface of the Heat Transfer Module with an interface of another module (CFD Module, AC/DC Module).

The multiphysics interfaces are found under the Heat Transfer branch ( ), and their availability depends on the COMSOL products available.

	For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

The multiphysics interfaces add Domain Multiphysics Couplings and Boundary Multiphysics Couplings. They predefine the couplings through specific settings in the multiphysics couplings and in the constituent interfaces to facilitate easy set up of models. These settings are detailed in the following sections:

- The Non-Isothermal Flow and Conjugate Heat Transfer Interfaces
- The Thermoelectric Effect Interface
- The Local Thermal Non-Equilibrium Interface
- The Heat and Moisture Transport Interface

A brief description of other multiphysics interfaces coupling an interface of the Heat Transfer Module with other interfaces is given in the following sections:

- The Joule Heating Interface
- The Laser Heating Interface
- The Induction Heating Interface
- The Microwave Heating Interface

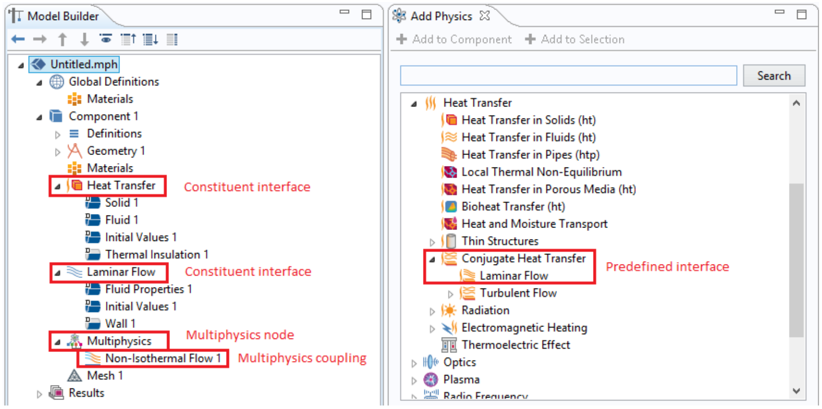
Links to thorough information about these interfaces are given in the corresponding sections.

Also:

- See The Thermal Stress Interface and The Joule Heating and Thermal Expansion Interface in the *Structural Mechanics Module User's Guide* for other multiphysics interfaces having The Heat Transfer in Solids Interface as a constituent interface. These two multiphysics interfaces (found under the Structural Mechanics branch ( )) add in particular the Thermal Expansion multiphysics coupling.
- See each constituent interface documentation for more details about the common settings (in The Heat Transfer Module Interfaces for the Heat Transfer Module interfaces, and in the other modules documentation for the other interfaces).
- See The Heat Transfer Features for details about the features available with the Heat Transfer Module constituent interfaces.
- See Multiphysics Couplings for details about the multiphysics couplings added with the Heat Transfer Module predefined multiphysics interfaces.

The predefined interfaces are found under the branches of the **Model Wizard** or **Add Physics** windows. They add the constituent interfaces and the **Multiphysics** node containing one or more multiphysics couplings.

On the figure below, the predefined interface **Conjugate Heat Transfer, Laminar Flow** is found under the **Heat Transfer** branch. It adds the constituent interfaces **Heat Transfer** and **Laminar Flow**, and the **Multiphysics** node containing the **Non-Isothermal Flow** multiphysics coupling.



The Non-Isothermal Flow and Conjugate Heat Transfer Interfaces

In this section:

- Advantages of Using the Multiphysics Interfaces
- The Non-Isothermal Flow, Laminar Flow and Turbulent Flow Interfaces
- The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces
- Settings for Physics Interfaces and Coupling Features
- Coupling Features
- Physics Interface Features

See The Laminar Flow Interface, The Turbulent Flow, Algebraic yPlus Interface, The Turbulent Flow, L-VEL Interface, The Turbulent Flow, k-e Interface, and The Turbulent Flow, Low Re k-e Interface in the *CFD Module User's Guide* for a description of the laminar and turbulent single-phase flow interfaces.

See Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide* for a description of the nodes associated to these interfaces.

Advantages of Using the Multiphysics Interfaces

The Non-Isothermal Flow and Conjugate Heat Transfer interfaces combine the heat equation with either laminar flow or turbulent flow and use either a **Fluid**, **Porous Medium**, or **Phase Change Material** domain model. The advantage of using the multiphysics interfaces is that predefined couplings are available in both directions. In particular, physics interfaces use the same definition of the density, which can therefore be a function of both pressure and temperature. Solving this coupled system of equations usually requires numerical stabilization accounting for the couplings, which the predefined multiphysics interfaces also set up.

When the flow **Compressibility** is set to **Incompressible flow**, the **Non-Isothermal Flow** coupling follows the Boussinesq approximation and evaluates the thermal material properties at the reference temperature. Hence, the **Phase Change Material** domain model—which requires the temperature dependency of the heat capacity—should be used only with the **Weakly compressible flow** and **Compressible flow (Ma<0.3)** options.

When the **Non-Isothermal Flow** and **Conjugate Heat Transfer** interfaces couple a turbulent flow with a **Phase Change Material** domain model, it must be noted that only the geometry boundaries are considered for the definition of the wall functions and of the wall distance. For example, at the moving melting front between the solid and the liquid phase, no wall functions are used by the k-ε and k-ω turbulence models.

See also Heat Transfer and Fluid Flow Coupling for more information.

The Non-Isothermal Flow, Laminar Flow and Turbulent Flow Interfaces

When a multiphysics interface is added from the **Fluid Flow>Non-Isothermal Flow** branch of the **Model Wizard** or **Add Physics** windows, one of the **Single-Phase Flow** interfaces (laminar or turbulent flow) and **Heat Transfer** are added to the Model Builder.

In addition, the **Multiphysics** node is added, which includes the multiphysics coupling feature **Non-Isothermal Flow**.

The Multiphysics Node in the *COMSOL Multiphysics Reference Manual.*

- The **Laminar Flow** interface ( ) combines a Heat Transfer in Fluids interface with a Laminar Flow interface.

- The **Turbulent Flow, Algebraic yPlus** interface ( ) combines a Heat Transfer in Fluids interface with a Turbulent Flow, Algebraic yPlus interface.
- The **Turbulent Flow, L-VEL** interface ( ) combines a Heat Transfer in Fluids interface with a Turbulent Flow, L-VEL interface. The **Turbulent Flow, k-ε** interface ( ) combines a Heat Transfer in Fluids interface with a Turbulent Flow, k-ε interface.
- The **Turbulent Flow, Low Re k-ε** interface ( ) combines a Heat Transfer in Fluids interface with a Turbulent Flow, Low Re k-ε interface.

The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces

When a multiphysics interface is added from the **Heat Transfer>Conjugate Heat Transfer** branch of the **Model Wizard** or **Add Physics** windows, **Heat Transfer** and one of the **Single-Phase Flow** interfaces (laminar or turbulent flow) are added to the Model Builder.

In addition, the **Multiphysics** node is added, which includes the multiphysics coupling feature **Non-Isothermal Flow**.

The Multiphysics Node in the *COMSOL Multiphysics Reference Manual*.

- The **Laminar Flow** interface ( ) combines a Heat Transfer interface with a Laminar Flow interface.
- The **Turbulent Flow, Algebraic yPlus** interface ( ) combines a Heat Transfer interface with a Turbulent Flow, Algebraic yPlus interface.
- The **Turbulent Flow, L-VEL** interface ( ) combines a Heat Transfer interface with a Turbulent Flow, L-VEL interface. The **Turbulent Flow, k-ε** interface ( ) combines a Heat Transfer interface with a Turbulent Flow, k-ε interface.
- The **Turbulent Flow, Low Re k-ε** interface ( ) combines a Heat Transfer interface with a Turbulent Flow, Low Re k-ε interface.

Settings for Physics Interfaces and Coupling Features

When physics interfaces are added using the predefined couplings, specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Heat Transfer in Fluids** and **Laminar Flow** interfaces are added, COMSOL adds an empty **Multiphysics** node. When you right-click this node, you can choose from the available coupling features — **Non-Isothermal Flow**, **Temperature Coupling**, and **Flow Coupling** — but the modified settings are not included.

TABLE 8-1: MODIFIED SETTINGS FOR THE NON-ISOTHERMAL FLOW INTERFACES

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Solids	Discretization order from temperature Lagrange shape function is 1. A **Fluid** feature is added with a empty default editable selection.
Heat Transfer in Fluids	none
Non-Isothermal Flow	The **Fluid flow** and **Heat transfer** interfaces are preselected

Coupling Features

See Non-Isothermal Flow, Flow Coupling, and Temperature Coupling for a description of the multiphysics couplings.

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using.

HEAT TRANSFER IN FLUIDS (NON-ISOTHERMAL FLOW)

The available physics features for The Heat Transfer in Fluids Interface are listed in Feature Nodes for the Heat Transfer in Fluids Interface. Also see Fluid for details about that feature.

HEAT TRANSFER IN SOLIDS (CONJUGATE HEAT TRANSFER)

The available physics features for The Heat Transfer in Solids Interface are listed in Feature Nodes for the Heat Transfer in Solids Interface. Also see Solid for details about that feature.

LAMINAR FLOW

The available physics features for The Laminar Flow Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

TURBULENT FLOW, ALGEBRAIC YPLUS

The available physics features for The Turbulent Flow, Algebraic yPlus Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

TURBULENT FLOW, L-VEL

The available physics features for The Turbulent Flow, L-VEL Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

TURBULENT FLOW, k-ε

The available physics features for The Turbulent Flow, k-e Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

TURBULENT FLOW, LOW RE k-ε

The available physics features for The Turbulent Flow, Low Re k-e Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

The Thermoelectric Effect Interface

The Thermoelectric Effect Interface

When the predefined **Thermoelectric Effect** ( ) interface is added (found under the **Heat Transfer** branch ( ) of the **Model Wizard** or **Add Physics** windows), it combines the Electric Currents and the Heat Transfer in Solids interfaces for modeling Peltier-Seebeck-Thomson effects. In addition, the **Electromagnetic Heat Source** and **Thermoelectric Effect** multiphysics couplings are added automatically.

The multiphysics couplings add the thermoelectric effect, the electromagnetic power dissipation, and the electromagnetic material properties, which can depend on the temperature.

Depending on the product license, stationary modeling and time-domain modeling are supported in all space dimensions. In addition, combinations of frequency-domain modeling for the Electric Currents interface and stationary modeling for the Heat Transfer in Solids interface, called frequency-stationary and frequency-transient modeling, are supported.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Electric Currents** and **Heat Transfer in Solids** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which includes the multiphysics coupling features **Thermoelectric Effect**, **Electromagnetic Heat Source**, **Boundary Thermoelectric Effect**, **Boundary Electromagnetic Heat Source**, and **Temperature Coupling**.

The Electric Currents interface calculates the electric field, current, and potential distributions in conducting media under conditions where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. Depending on the licensed products, time and frequency domain formulations that account for capacitive effects are also provided. The Electric Currents interface solves a current conservation equation based on Ohm's law using the scalar electric potential as the dependent variable.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A Solid model is active by default on all domains. All functionality for including other domain types, like a fluid domain, is also available. The temperature equation defined in solid domains corresponds to the differential form of the Fourier's law that may contain additional contributions like heat sources.

AS AN ADD-ON MULTIPHYSICS COUPLING

The Thermoelectric Effect multiphysics coupling is also available when there is at least one compatible Heat Transfer interface and one compatible AC/DC interface.

The compatible Heat Transfer interfaces are:

- Heat Transfer in Solids and Heat Transfer in Fluids
- Heat Transfer in Biological Tissue
- Heat Transfer in Porous Media
- Heat Transfer with Surface-to-Surface Radiation and Heat Transfer with Radiation in Participating Media
- Heat Transfer in Thin Shells

The compatible AC/DC interfaces are:

- Electric Currents and Electric Currents, Shell
- Magnetic Fields, Magnetic Field Formulation, and Magnetic and Electric Fields
- Rotating Machinery, Magnetic

Settings for Physics Interfaces and Coupling Features

When physics interfaces are added using the predefined couplings — for example, **Thermoelectric Effect** — specific settings are included with the physics interfaces and the coupling features.

However, if physics interfaces are added one at a time, followed by the coupling features, these modified settings are not automatically included.

For example, if single **Electric Currents** and **Heat Transfer in Solids** interfaces are added, COMSOL adds an empty **Multiphysics** node. You can choose from the following available coupling features: **Thermoelectric Effect**, **Electromagnetic Heat Source**, **Boundary Thermoelectric Effect**, **Boundary Electromagnetic Heat Source**, and **Temperature Coupling**, but the modified settings are not included.

Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

TABLE 8-2: MODIFIED SETTINGS FOR A THERMOELECTRIC EFFECT INTERFACE

PHYSICS INTERFACE	MODIFIED SETTINGS
Electric Currents	No changes.
Heat Transfer in Solids	No changes.
Thermoelectric Effect	The **Domain Selection** is the same as that of the participating physics interfaces. The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Thermoelectric Effect section.
Electromagnetic Heat Source	The **Domain Selection** is the same as that of the participating physics interfaces. The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Electromagnetic Heat Source section.
Boundary Thermoelectric Effect	The **Boundary Selection** is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces. The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Boundary Thermoelectric Effect section.
Boundary Electromagnetic Heat Source	The **Domain Selection** is the same as that of the participating physics interfaces. The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Boundary Electromagnetic Heat Source section.
Temperature Coupling	The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Temperature Coupling section.

Coupling Features

See Thermoelectric Effect, Boundary Thermoelectric Effect, Electromagnetic Heat Source, Boundary Electromagnetic Heat Source, and Temperature Coupling for a description of the multiphysics couplings.

Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

> In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

HEAT TRANSFER IN SOLIDS

The available physics features for The Heat Transfer in Solids Interface are listed in Feature Nodes for the Heat Transfer in Solids Interface.

ELECTRIC CURRENTS

The available physics features for The Electric Currents Interface are listed in Domain, Boundary, Edge, Point, and Pair Nodes for the Electric Currents Interface in the *COMSOL Multiphysics Reference Manual*.

The Local Thermal Non-Equilibrium Interface

The Local Thermal Non-Equilibrium Interface

When the predefined **Local Thermal Non-Equilibrium**( ) interface is added (found under the **Heat Transfer** branch ( ) of the **Model Wizard** or **Add Physics** windows), it combines the Heat Transfer in Solids and the Heat Transfer in Fluids interfaces to model heat transfer in porous media for which the solid and fluid temperatures are not in equilibrium.

Stationary and time-domain modeling are supported in all space dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Solids** and **Heat Transfer in Fluids** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Local Thermal Non-Equilibrium**.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A **Solid** model is active by default on all domains.

The Heat Transfer in Fluids interface provides features for modeling heat transfer by conduction, convection, and radiation. A **Fluid** model is active by default on all domains.

AS AN ADD-ON MULTIPHYSICS COUPLING

The Local Thermal Non-Equilibrium multiphysics coupling is also available when there is at least one of each of the following interfaces with the default model activated:

- Heat Transfer in Solids, with Solid model
- Heat Transfer in Fluids, with Fluid model

Coupling Feature

See Local Thermal Non-Equilibrium for details about the multiphysics coupling feature.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

HEAT TRANSFER IN SOLIDS

The available physics features for The Heat Transfer in Solids Interface are listed in Feature Nodes for the Heat Transfer in Solids Interface.

HEAT TRANSFER IN FLUIDS

The available physics features for The Heat Transfer in Fluids Interface are listed in Feature Nodes for the Heat Transfer in Fluids Interface.

Heat sources

In the heat source features available for the constituent interfaces (Heat Source, Geothermal Heating, and Bioheat), the user input corresponds to the heat production per total unit volume. It is multiplied by the volume fraction of each phase and added into the corresponding heat equation.

Heat fluxes

In the heat flux features available for the constituent interfaces (Heat Flux, Inflow Heat Flux), the user input corresponds to the heat flux per total unit surface. It is multiplied by the volume fraction of each phase and added into the corresponding heat equation. The surface fraction is approximated by the volume fraction.

The Heat and Moisture Transport Interface

The Heat and Moisture Transport Interface

When the predefined **Heat and Moisture Transport** ( ) interface is added (found under the **Heat Transfer** branch ( ) of the **Model Wizard** or **Add Physics** windows), it combines the Heat Transfer in Building Materials and the Moisture Transport interfaces to model coupled heat and moisture transport in building materials, by taking into account heat and moisture storage, latent heat effects, and liquid and convective transport of moisture.

Stationary and time-domain modeling are supported in all space dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Building Materials** and **Moisture Transport** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the **Heat and Moisture** multiphysics coupling feature.

The Heat Transfer in Building Materials interface provides features for modeling heat transfer by conduction, convection, and radiation. The **Building Material** model, active by default in all domains, provides in addition the functionality for moisture content dependency of thermodynamics properties and latent heat effects.

The Moisture Transport interface provides features for modeling moisture transfer by liquid transport (capillary flow) and vapor diffusion. A **Porous Medium** model is active by default on all domains.

AS AN ADD-ON MULTIPHYSICS COUPLING

The **Heat and Moisture** multiphysics coupling is also available when there is at least one of each of the following interfaces with the specified model activated:

- Any version of the Heat Transfer interface, with **Building Material** model
- Moisture Transport interface, with **Porous Medium** model

Coupling Feature

See Heat and Moisture for details about the multiphysics coupling feature.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).

In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

HEAT TRANSFER IN BUILDING MATERIALS

The available physics features for The Heat Transfer in Building Materials Interface are the same as for the The Heat Transfer in Porous Media Interface and are listed in Feature Nodes for the Heat Transfer in Porous Media Interface.

MOISTURE TRANSPORT

The available physics features for The Moisture Transport Interface are listed in Feature Nodes for the Moisture Transport Interface.

The Joule Heating Interface

In this section:

- The Joule Heating Interface
- Coupling Features

The Joule Heating Interface

The **Joule Heating** ( ) interface is used to model resistive heating and, depending on additional licensed products, dielectric heating in devices where inductive effects are negligible; that is, when the skin depth is much larger than the studied device. This multiphysics interface adds an Electric Currents interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic power dissipation as a heat source, and the electromagnetic material properties can depend on the temperature.

See The Joule Heating Interface in the *COMSOL Multiphysics Reference Manual* for more details about this multiphysics interface.

Coupling Features

See Electromagnetic Heat Source, Boundary Electromagnetic Heat Source, and Temperature Coupling for a description of the multiphysics couplings.

The Laser Heating Interface

In this section:

- The Laser Heating Interface
- Coupling Features

The Laser Heating Interface

The **Laser Heating** ( ) interface is used to model electromagnetic heating for systems and devices where the electric field amplitude varies slowly on a wavelength scale. This multiphysics interface adds an Electromagnetic Waves, Beam Envelopes interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic losses from the electromagnetic waves as a heat source, and the electromagnetic material properties can depend on the temperature. The modeling approach is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale.

See The Laser Heating Interface in the *Wave Optics Module User's Guide* for more details about this multiphysics interface.

Coupling Features

See Electromagnetic Heat Source, Boundary Electromagnetic Heat Source, and Temperature Coupling for a description of the multiphysics couplings.

The Induction Heating Interface

In this section:

- The Induction Heating Interface
- Coupling Features

The Induction Heating Interface

The **Induction Heating** interface ( ) is used to model induction heating and eddy current heating. This multiphysics interface adds a Magnetic Fields interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic power dissipation as a heat source, and the electromagnetic material properties can depend on the temperature.

See The Induction Heating Interface in the *AC/DC Module User's Guide* for more details about this multiphysics interface.

Coupling Features

See Electromagnetic Heat Source, Boundary Electromagnetic Heat Source, and Temperature Coupling for a description of the multiphysics couplings.

The Microwave Heating Interface

In this section:

- The Microwave Heating Interface
- Coupling Features

The Microwave Heating Interface

The **Microwave Heating** interface () is used to model electromagnetic heating for systems and devices that are on a scale ranging from 1/10 of a wavelength up to, depending on available computer memory, about 10 wavelengths. This multiphysics interface adds an Electromagnetic Waves, Frequency Domain interface and a Heat Transfer in Solids interface. The multiphysics couplings add the electromagnetic losses from the electromagnetic waves as a heat source, and the electromagnetic material properties can depend on the temperature. The modeling approach is based on the assumption that the electromagnetic cycle time is short compared to the thermal time scale.

See The Microwave Heating Interface in the *RF Module User's Guide* for more details about this multiphysics interface.

Coupling Features

See Electromagnetic Heat Source, Boundary Electromagnetic Heat Source, and Temperature Coupling for a description of the multiphysics couplings.

9

Multiphysics Couplings

The Heat Transfer Module has multiphysics couplings available under certain conditions.

When a predefined multiphysics interface is added from the **Model Wizard** or **Add Physics** windows, it adds the constituent interfaces and the **Multiphysics** node, which automatically includes one or more multiphysics couplings.

If the constituent physics interfaces are added one at a time, then it adds an empty **Multiphysics** node. When you right-click this node, you can choose from the available multiphysics couplings.

The Multiphysics Node in the *COMSOL Multiphysics Reference Manual*.

The default settings of the couplings depend on the way the **Multiphysics** node was created.

In this chapter, the following multiphysics couplings are described:

- Domain Multiphysics Couplings
- Boundary Multiphysics Couplings

See The Heat Transfer Module Interfaces for details about the Heat Transfer Module interfaces.

See Multiphysics Interfaces for details about the predefined multiphysics interfaces of the Heat Transfer Module.

Domain Multiphysics Couplings

The Heat Transfer Module has the following domain multiphysics couplings available:

- Electromagnetic Heat Source
- Flow Coupling
- Heat and Moisture
- Local Thermal Non-Equilibrium
- Non-Isothermal Flow
- Temperature Coupling
- Thermal Expansion
- Thermoelectric Effect

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Electromagnetic Heat Source

Use this multiphysics coupling ( ) to account for electromagnetic losses in the heat equation.

See Electromagnetic Heat Source in the *COMSOL Multiphysics Reference Manual* for a description of this multiphysics coupling in the context of Joule heating modeling.

See Electromagnetic Heat Source in the *AC/DC Module User's Guide* for a description of this multiphysics coupling in the context of induction heating modeling.

See Electromagnetic Heat Source in the *RF Module User's Guide* for a description of this multiphysics coupling in the context of microwave heating modeling.

Hepatic Tumor Ablation: Application Library path **Heat_Transfer_Module/Medical_Technology/tumor_ablation**

Flow Coupling

This multiphysics coupling ( ) defines **u** and *p* variables in order to set the model inputs in the Heat Transfer interface (or when applicable, a chemical species transport interface). In addition it provides all the fluids quantities that may be needed by the Heat Transfer interface (for example, viscosity, turbulence parameters).

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `fc1`.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. By default, the applicable physics interface is selected in the **Source** list to apply the **Destination** to its physics interface to establish the coupling.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Heat Transfer in Solids** is deleted — then the **Destination** list defaults to **None** as there is nothing to couple to.

> If a physics interface is deleted and then added to the model again, and in order to re-establish the coupling, you need to choose the physics interface again from the lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present physics interface. See Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual.*

LOCATION IN USER INTERFACE

Context menus

Multiphysics>Flow Coupling

when any of the following interface is added together with **Heat Transfer in Solids** (or another version of the Heat Transfer Interface):

Single-Phase Flow (any version)

Porous Media and Subsurface Flow, Brinkman Equations

Heat and Moisture

Use this multiphysics coupling ( ) to model coupled heat and moisture transfer in building materials, by taking into account heat and moisture storage, latent heat effects, and liquid and convective transport of moisture. The heat and moisture coupling can be applied to the computation of different moisture variations phenomena in building components, such as drying of initial construction moisture, condensation due to migration of moisture from outside to inside in summer, and moisture accumulation by interstitial condensation due to diffusion in the winter.

The thermodynamics properties of the building material depend both on the dry solid properties and on the moisture content, and the evaporation of liquid water adds a latent heat source in the diffusion equation for temperature. Reversely, the variations of moisture content due to liquid transport (capillary flow) and vapor diffusion are temperature dependent.

The **Heat and Moisture** coupling synchronizes the features from the **Heat Transfer** and **Moisture transport** interfaces:

- It defines the relative humidity ϕ in order to set the model inputs in the features of the **Heat Transfer** interface.
- It defines the moisture storage function w and the vapor permeability δ_{p} (or vapor resistance factor μ) in order to set the corresponding inputs in the **Building Material** feature of the **Heat Transfer** interface.
- It defines the temperature T in order to set the model inputs in the features of the **Moisture transport** interface.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `ham1`.

DOMAIN SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific domains to define the domains with heat and moisture transport, or select **All domains** as needed.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling.

Select the **Heat Transfer** interface associated to the temperature dependent variable. Select the **Moisture transport** interface associated to the relative humidity variable.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Heat Transfer in Building Materials** is deleted — then the **Heat transfer** list defaults to **None** as there is nothing to couple to.

LOCATION IN USER INTERFACE

Context menus

Multiphysics>Heat and Moisture

when any version of the heat transfer interface with **Building Material** feature is added together with the **Moisture Transport** interface.

Local Thermal Non-Equilibrium

Use this multiphysics coupling ( ) to account for heat transfer in porous domains where the solid and fluid temperatures are not in equilibrium. This is achieved by coupling the heat equations in the solid and fluid subdomains through a transfer term proportional to the temperature difference between the fluid and the solid. The corresponding heat equations in the solid and in the fluid subdomains read

$$\theta_p\rho_s C_{p,\,s}\frac{\partial T_s}{\partial t}+\theta_p\rho_s C_{p,\,s}\mathbf{u}_s\cdot\nabla T_s = \nabla\cdot(\theta_p k_s\nabla T_s)+q_{sf}(T_f-T_s)$$

$$(1-\theta_p)\rho_f C_{p,\,f}\frac{\partial T_f}{\partial t}+(1-\theta_p)\rho_f C_{p,\,f}\mathbf{u}_f\cdot\nabla T_f = \nabla\cdot((1-\theta_p)k_f\nabla T_f)+q_{sf}(T_s-T_f)$$

with the following material properties:

- θ_p is the solid volume fraction.
- ρ_s and ρ_f are the solid and fluid densities.

- $C_{p,\mathrm{s}}$ and $C_{p,\mathrm{f}}$ are the solid and fluid heat capacities at constant pressure.
- k_s and k_f are the solid and fluid thermal conductivities.
- q_sf is the interstitial convective heat transfer coefficient.
- $\mathbf{u}_\mathrm{s}$ and $\mathbf{u}_\mathrm{f}$ are the solid and fluid velocity vectors.

The fluid velocity is deduced from a porous velocity $\mathbf{u}_\mathrm{p}$, coming, for example, from the Darcy's law or the Brinkmann equations, according to:

$$\mathbf{u}_\mathrm{f} = \frac{\mathbf{u}_\mathrm{p}}{1-\theta_\mathrm{p}}$$

See also Local Thermal Non-Equilibrium in Theory for Heat Transfer in Porous Media.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `ltne1`.

LOCAL THERMAL NON-EQUILIBRIUM SETTINGS

Enter a **Solid volume fraction** θ_p (dimensionless). The default value is 0.5.

Select an **Interstitial convective heat transfer coefficient**: **Spherical pellet bed**, **General configuration**, or **User defined** (the default).

Spherical Pellet Bed

In this particular configuration, the interstitial convective heat transfer coefficient can be directly expressed as a function of the average pellet radius r_p and the fluid-to-solid Nusselt number for which the fluid dynamic viscosity μ is needed.

Enter a value for the **Average pellet radius** r_p (SI unit: m). Default value is 5e-4 m.

The default **Dynamic viscosity** μ (SI unit: Pa·s) is used **From material**. In the list, choose **User defined** to enter another value or expression. When the dynamic viscosity is set in the Heat Transfer in Fluids interface, it also appears in the list.

> The Heat Transfer in Fluids interface defines the dynamic viscosity if either **Moist air** is selected as **Fluid type** in the **Thermodynamics, Fluid** section, or the **Equivalent conductivity for convection** check box is selected in the **Equivalent Conductivity for Convection** section.

General Configuration

The interstitial convective heat transfer coefficient is expressed as the product of the specific surface area a_{sf} and the interstitial heat transfer coefficient h_{sf}.

Enter a value for the **Specific surface area** a_{sf} (SI unit: 1/m). Default value is 0 1/m.

Enter a value for the **Interstitial heat transfer coefficient** h_{sf} (SI unit: W/(m^2·K)). Default value is 0 W/(m^2·K).

User Defined

Enter a custom value for q_{sf} (SI unit: W/(m^3·K)). Default value is 0 W/(m^3·K).

COUPLED INTERFACES

This section defines the physics involved in the **Local Thermal Non-Equilibrium** multiphysics coupling.

Select the **Heat transfer in solids** interface associated to the solid temperature dependent variable. Select the **Heat transfer in fluids** interface associated to the fluid temperature dependent variable.

LOCATION IN USER INTERFACE

Context menus

Multiphysics>Local Thermal Non-Equilibrium

when the **Heat Transfer in Solids** interface with **Solid** feature is added together with the **Heat Transfer in Fluids** interface with **Heat Transfer in Fluids** feature.

Non-Isothermal Flow

Use this multiphysics coupling to simulate fluid flows where the fluid properties depend on temperature. Models can also include heat transfer in solids or in porous media as well as surface-to-surface radiation and radiation in participating media, with the Heat Transfer Module. The physics interface supports low Mach numbers (typically less than 0.3).

The Non-Isothermal Flow, Laminar Flow interface solves for conservation of energy, mass and momentum in fluids and porous media and for conservation of energy in solids. It synchronizes the features from the **Heat Transfer** and **Fluid Flow** interfaces when a turbulent flow regime is defined. It also complements the **Screen** and **Interior Fan** feature from the flow interface to account for thermal effects. The physics interface can be used for stationary and time-dependent analysis.

When the **Non-Isothermal Flow** is used, there is no need to add **Flow Coupling** or **Temperature Coupling**. Indeed, **Non-Isothermal Flow** combines the effects of both of them. In addition it also accounts for the multiphysics stabilization terms and for the heat transfer changes in the turbulent regime (for example, thermal wall functions).

The multiphysics stabilizations (streamline diffusion and crosswind diffusion) are controlled by the Fluid Flow interface. For example, the multiphysics streamline diffusion can be disabled in a **Laminar Flow** physics node, in the **Stabilization** section. The stabilization selected in the Heat Transfer physics interface has no effect if the multiphysics coupling stabilization is active, but remains active if not. However, the isotropic diffusion is not a multiphysics stabilization and is controlled by each physics interface.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is nitf1.

DOMAIN SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific domains to define the non-isothermal flow, or select **All domains** as needed.

MATERIAL PROPERTIES

Select an option from the **Specify density** list: **From heat transfer interface** (the default), **From fluid flow interface**, or **Custom**. For **Custom**, enter a **Density** ρ (SI unit: kg/m^3), or select a density in the list if available.

HEAT TURBULENCE PROPERTY

Select an option from the **Heat transport turbulence model** list: **Kays-Crawford** (the default), **Extended Kays-Crawford**, or **User-defined turbulent Prandtl number**.

For **Extended Kays-Crawford**, enter a **Reynolds number at infinity** $\mathrm{Re}_{\mathrm{inf}}$ (dimensionless).

For **User-defined turbulent Prandtl number**, enter a **Turbulent Prandtl number** pr_{T} (dimensionless).

When the flow interface uses a RANS turbulence model, the conductive heat flux is defined as

$$q = -(k + k_{\mathrm{T}})\nabla T$$

with the turbulent thermal conductivity defined as

$$k_{\mathrm{T}} = \frac{\mu_{\mathrm{T}} C_p}{Pr_{\mathrm{T}}}$$

where μ_{T} is defined by the flow interface, and Pr_{T} depends on the **Heat transport turbulence model**. See Turbulent Conductivity for details.

FLOW HEATING

Select the **Include work done by pressure changes** check box to account for the heat source due to pressure changes:

$$Q_p = \alpha_p \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right)$$

By default this option is not selected; however, it should be selected for compressible fluids as soon as significant pressure gradients occur.

Select the **Include viscous dissipation** check box to account for the heat source corresponding to viscous heating. This option is not selected by default. Because it may induce an extra computational cost it should be only selected in application where such effect is expected. If no information on this is available, selecting the option ensures that the energy balance for the heat and the flow equation is respected.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Fluid flow** and **Heat transfer** lists include all applicable physics interfaces.

The default values depend on how this coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Heat Transfer in Fluids** is deleted — then the **Heat transfer** list defaults to **None** as there is nothing to couple to.

When an interface is selected from the **Heat transfer** list, some of its model inputs are forced with values from the **Non-Isothermal Flow** node. In addition, it defines how the turbulence has to be accounted for, depending on the **Fluid flow** interface's turbulence settings. Therefore, each heat transfer or fluid flow interface should be used in at most one **Non-Isothermal Flow** node. In cases where multiple fluid flow interfaces are used, an equal number of heat transfer interfaces and **Non-Isothermal Flow** nodes are needed to define proper multiphysics couplings.

If a physics interface is deleted and then added to the model again, then in order to re-establish the coupling, you need to choose the physics interface again from the **Fluid flow** or **Heat transfer** lists. This behavior is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

Heat Sink: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/heat_sink**

LOCATION IN USER INTERFACE

Context menus

Multiphysics>Non-Isothermal Flow

when any of the following interface is added together with **Heat Transfer in Solids** (or another version of the Heat Transfer Interface):

Single-Phase Flow (any version)

Porous Media and Subsurface Flow, Brinkman Equations

Temperature Coupling

Use this multiphysics coupling to add the temperature as the default model input for a standalone physics interface.

COUPLED INTERFACES

The **Temperature Coupling** feature is generic and specifies a Heat Transfer interface as **Source** and a second interface as **Destination**. When **Temperature Coupling** feature is used, the temperature from the **Source** is used to evaluate material properties in any feature from the **Destination** interface. The coupling can be added wherever the Heat Transfer interface is active.

The **Source** interface can be any interface defining a temperature, which includes all versions of heat transfer and multiphysics, except the pure radiation interfaces.

The **Destination** interface can be any interface providing multiphysics feature in the **Multiphysics** node — for example, **Electric Current** or **Solid Mechanics**.

See Temperature Coupling in the *COMSOL Multiphysics Reference Manual* for more details about this multiphysics coupling.

	Contact Switch: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/contact_switch**

Thermal Expansion

Use this multiphysics coupling ( ) to add an internal thermal strain caused by changes in temperature.

See Thermal Expansion (Multiphysics Coupling) in the *Structural Mechanics Module User's Guide* for more details about this multiphysics coupling.

	Heating Circuit: Application Library path **Heat_Transfer_Module/Power_Electronics_and_Electronic_Circuit/heating_circuit**

Thermoelectric Effect

Use this multiphysics coupling ( ) to account for a Peltier heat source or sink in domains where electrical and thermal models are defined. This modeling is achieved by adding $P\mathbf{J}$ contribution to the heat flux. The corresponding heat equation in an immobile solid reads

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

A **Thermoelectric Effect** node also adds the term $-\sigma S\nabla T$ to the current density, which is then defined as:

$$\mathbf{J} = -\sigma(\nabla V + S\nabla T)$$

	The contribution to the heat flux affects all boundary conditions where the conductive flux, $-k\nabla T$, is involved. In particular the thermal insulation condition becomes $(-k\nabla T + P\mathbf{J}) \cdot \mathbf{n} = 0$ (instead of $(-k\nabla T) \cdot \mathbf{n} = 0$ when thermoelectric effect is not active).

In 2D, the thickness used in the multiphysics contributions is the one defined in the electromagnetic interface.

Theory for the Thermoelectric Effect Interface

Thermoelectric Leg: Application Library path **Heat_Transfer_Module/Verification_Examples/thermoelectric_leg**

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `tee`.

THERMOELECTRIC PROPERTIES

The default Seebeck coefficient S (SI unit: V/K) is taken **From material**. For **User defined** enter other values or expressions.

COUPLED INTERFACES

This section defines the physics involved in the thermoelectric effect multiphysics coupling.

Select the **Heat Transfer** interface associated to the temperature dependent variable. Select the **Electromagnetic** interface associated to the electric potential dependent variable.

LOCATION IN USER INTERFACE

Context menus

Multiphysics>Thermoelectric Effect

when any of the following interface is added together with **Heat Transfer in Solids** (or another version of the Heat Transfer Interface):

Electric Currents

Magnetic Field Formulation

Magnetic Fields

Magnetic and Electric Fields

Rotating Machinery, Magnetic

Boundary Multiphysics Couplings

The Heat Transfer Module has the following boundary multiphysics couplings available:

- Boundary Thermoelectric Effect
- Boundary Electromagnetic Heat Source
- Marangoni Effect

For a detailed overview of the functionality available in each product, visit http://www.comsol.com/products/specifications/

Boundary Thermoelectric Effect

Use this multiphysics coupling ( ) to account for a Peltier heat source or sink on boundaries where electric and thermal shells are defined. This modeling is achieved by adding $P_s\mathbf{J}_s$ contribution to the heat flux. The corresponding heat equation in an immobile solid reads:

$$d_s\rho_s C_s\frac{\partial T}{\partial t} + \nabla_{\mathbf{t}}\cdot d_s(-k_s\nabla_{\mathbf{t}}T + P_s\mathbf{J}_s) = d_s Q_s$$

A **Boundary Thermoelectric Effect** node also contributes the term $-d_s\sigma_s S_s\nabla_{\mathbf{t}}T$ to the current density, which is then defined as:

$$\mathbf{J}_s = -d_s\sigma_s(\nabla_{\mathbf{t}}V + S_s\nabla_{\mathbf{t}}T)$$

In 2D, the thickness used the multiphysics contributions is the one defined in the electromagnetic interface.

Theory for the Thermoelectric Effect Interface

Thermoelectric Leg: Application Library path
Heat_Transfer_Module/Verification_Examples/thermoelectric_leg

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `btee`.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries where boundary thermoelectric effect is defined. Only boundaries where both **Thin Layer** and **Electric Shielding** are active can be selected.

The weak contribution in only added when **Layer type** in **Thin Layer** is set to **Conductive**. If **Layer type** is set to either **Resistive** or **General**, the weak contribution evaluates to 0.

THERMOELECTRIC PROPERTIES

The default Seebeck coefficient S (SI unit: V/K) is taken **From material**. For **User defined** enter other values or expressions.

BOUNDARY THERMOELECTRIC EFFECT

This section defines the physics involved in the thermoelectric effect multiphysics coupling.

Select the **Heat transfer** interface associated to the temperature dependent variable. Select the **Electromagnetic** interface associated to the electric potential dependent variable.

LOCATION IN USER INTERFACE

Context menus

Multiphysics>Boundary Thermoelectric Effect

when any of the following interface is added together with **Heat Transfer in Solids** (or another version of the Heat Transfer Interface):

Electric Currents

Electric Currents, Shells

Magnetic Field Formulation

Magnetic Fields

Magnetic and Electric Fields

Rotating Machinery, Magnetic

Boundary Electromagnetic Heat Source

This multiphysics coupling ( ) maps the electromagnetic surface losses as a heat source on the boundary in the heat transfer part of the model. It is a default node.

See Boundary Electromagnetic Heat Source in the *COMSOL Multiphysics Reference Manual* for a description of this multiphysics coupling.

Contact Switch: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/contact_switch**

Marangoni Effect

This multiphysics coupling ( ) accounts for Marangoni convection. Marangoni convection occurs when the surface tension of an interface (generally liquid-air) depends on the concentration of a species or on the temperature distribution. In the case of temperature dependence, the Marangoni effect is also called thermo-capillary convection. It is of primary importance in the fields of welding, crystal growth, and electron beam melting of metals.

The Marangoni effect is a shear stress which depends on the tangential temperature gradient and should be implemented as such. It has the following contribution described by forces induced on the fluid/fluid interface:

$$\left[-p\mathbf{I}+\mu(\nabla\mathbf{u}+(\nabla\mathbf{u})^{\mathrm{T}})-\frac{2}{3}\mu(\nabla\cdot\mathbf{u})\mathbf{I}\right]\mathbf{n} = \gamma\nabla_{\mathbf{t}}T$$

where γ is the temperature derivative of the surface tension σ (N/(m·K)). Note that this formulation is intended for laminar flow regimes only.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is me1.

SURFACE TENSION

Select a **Surface tension coefficient type**: **Library coefficient, liquid/gas interface** or **User defined** (the default).

- For **Library coefficient, liquid/gas interface** choose an option from the **Library surface tension coefficient** list.
- For **User defined** enter a **Surface tension coefficient** σ (SI unit: N/m).

MARANGONI EFFECT

This section defines the physics involved in the multiphysics coupling. The **Fluid flow** and **Heat transfer** lists include all applicable physics interfaces.

The default values depend on how this coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Heat Transfer in Fluids** is deleted — then the **Heat transfer** list defaults to **None** as there is nothing to couple to.

If a physics interface is deleted and then added to the model again, then in order to re-establish the coupling, you need to choose the physics interface again from the **Fluid flow** or **Heat transfer** lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Approaches in the *COMSOL Multiphysics Reference Manual*.

Marangoni Effect: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/marangoni_effect**

LOCATION IN USER INTERFACE

Context menus

Multiphysics>Marangoni Effect

when any of the following interface is added together with **Heat Transfer in Solids** (or another version of the Heat Transfer Interface):

Single-Phase Flow (any version)

Porous Media and Subsurface Flow, Brinkman Equations

Index