Heat Transfer Module

User's Guide

COMSOL

Heat Transfer Module User's Guide

Contact Information

Visit the Contact COMSOL page at www.comsol.com/contact to submit general inquiries 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/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- 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 6: The Heat Transfer Features

Boundary Features   **548**

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
- Additional 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 Nonisothermal Flow, Laminar Flow (nitf) and Nonisothermal Flow, Turbulent Flow (nitf) interfaces found under the **Fluid Flow>Nonisothermal 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 Nonisothermal 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				
Moisture Transport in Air		mt	all dimensions	stationary; time dependent
Moisture Transport in Porous Media		mt	all dimensions	stationary; time dependent
Moisture Transport in Building Materials		mt	all dimensions	stationary; time dependent
Moisture Flow				
Laminar Flow[2]		—	all dimensions	stationary; time dependent
Turbulent Flow				

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Turbulent Flow, Algebraic yPlus[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Porous Media Flow[2]		—	all dimensions	stationary; time dependent
≋ **Fluid Flow**				
≋ **Single-Phase Flow**				
Laminar Flow[1]	≋	spf	3D, 2D, 2D axisymmetric	stationary; time dependent
≋ *Turbulent Flow*				
Turbulent Flow, Algebraic yPlus		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε	≋	spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε	≋	spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
≋ **Nonisothermal Flow**				
Laminar Flow[1,2]	≋	—	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
≋ *Turbulent Flow*				
Turbulent Flow, Algebraic yPlus[2]	≋	—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, L-VEL[2]	≋	—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, k-ε[2]	≋	—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re k-ε[2]	≋	—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
⦚⦚⦚ **Heat Transfer**				
Heat Transfer in Solids[1]	▨	ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Heat Transfer in Fluids[1]	▨	ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Heat Transfer in Solids and Fluids[1]	▨	ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
▨ **Conjugate Heat Transfer**				
Laminar Flow[1,2]	▨	—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; thermal perturbation, frequency domain

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
≋ *Turbulent Flow*				
Turbulent Flow, Algebraic yPlus[2]	≋	—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization; stationary, one-way NITF; time dependent, one-way NITF
Turbulent Flow, L-VEL[2]	≋	—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization; stationary, one-way NITF; time dependent, one-way NITF
Turbulent Flow, k-ε[2]	≋	—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; thermal perturbation, frequency domain
Turbulent Flow, Low Re k-ε[2]	≋	—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; thermal perturbation, frequency domain
☀ **Radiation**				
Heat Transfer with Surface-to-Surface Radiation[2]	☀	—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Heat Transfer with Radiation in Participating Media[2]	☀▮	—	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain
Heat Transfer with Radiation in Absorbing-Scattering Media[2]	☀▮	—	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain
Heat Transfer with Radiative Beam in Absorbing Media[2]	🔥	—	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Surface-to-Surface Radiation		rad	all dimensions	stationary; time dependent
Radiation in Participating Media		rpm	3D, 2D, 2D axisymmetric	stationary; time dependent
Radiation in Absorbing-Scattering Media		rasm	3D, 2D, 2D axisymmetric	stationary; time dependent
Radiative Beam in Absorbing Media		rbam	3D, 2D, 2D axisymmetric	stationary; time dependent

Electromagnetic Heating

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Joule Heating[1,2]		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Lumped Thermal System		lts	all dimensions	stationary; time dependent

Thin Structures

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Heat Transfer in Shells		htlsh	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Heat Transfer in Films		htlsh	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Heat Transfer in Fractures		htlsh	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency

Heat and Moisture Transport

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Moist Air[2]		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Moist Porous Media[2]		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Building Materials[2]		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Heat and Moisture Flow				
Laminar Flow[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; one-way MF; time dependent, one-way MF; thermal perturbation, frequency domain
Turbulent Flow				
Turbulent Flow, Algebraic yPlus[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization; stationary, one-way NITF; time dependent, one-way NITF; one-way MF; time dependent, one-way MF
Turbulent Flow, L-VEL[2]		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization; stationary, one-way NITF; time dependent, one-way NITF; one-way MF; time dependent, one-way MF

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Turbulent Flow, k-ε[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; one-way MF; time dependent, one-way MF; thermal perturbation, frequency domain
Turbulent Flow, Low Re k-ε[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; one-way MF; time dependent, one-way MF; thermal perturbation, frequency domain
Porous Media Flow[2]		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; one-way MF; time dependent, one-way MF; thermal perturbation, frequency domain
🔴 **Porous Media**				
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Local Thermal Nonequilibrium		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Heat Transfer in Packed Beds		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Bioheat Transfer		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Thermoelectric Effect[2]		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain

[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 2-4 for links to common sections and Table 2-5 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					
			STATIONARY	TIME DEPENDENT	THERMAL PERTURBATION, EIGENFREQUENCY	THERMAL PERTURBATION, FREQUENCY DOMAIN	STATIONARY WITH INITIALIZATION	TRANSIENT WITH INITIALIZATION
CHEMICAL SPECIES TRANSPORT>MOISTURE TRANSPORT								
Moisture Transport in Air	mt	ϕ	√	√				
Moisture Transport in Porous Media	mt	ϕ	√	√				
Moisture Transport in Building Materials	mt	ϕ	√	√				
CHEMICAL SPECIES TRANSPORT>MOISTURE FLOW								
Laminar Flow[2]	—	u, p, ϕ	√	√				
Turbulent flow, Algebraic yPlus[2]	—	$u, p, G, yPlus, \phi$					√	√
Turbulent flow, L-VEL[2]	—	$u, p, G, uPlus, \phi$					√	√
Turbulent Flow, k-ε[2]	—	u, p, k, ep, ϕ	√	√				
Turbulent Flow, Low Re k-ε[2]	—	u, p, k, ep, G, ϕ					√	√
Porous Media Flow[2]	—	u, p, ϕ	√	√				
FLUID FLOW>SINGLE-PHASE FLOW								
Laminar Flow	spf	u, p	√	√				
Turbulent flow, Algebraic yPlus	spf	$u, p, G, yPlus$					√	√
Turbulent flow, L-VEL	spf	$u, p, G, uPlus$					√	√
Turbulent Flow, k-ε	spf	u, p, k, ep	√	√				

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

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	PRESET STUDIES					
			STATIONARY	TIME DEPENDENT	THERMAL PERTURBATION, EIGENFREQUENCY	THERMAL PERTURBATION, FREQUENCY DOMAIN	STATIONARY WITH INITIALIZATION	TRANSIENT WITH INITIALIZATION
Turbulent Flow, Low Re k-ε	spf	u, p, k, ep, G					√	√
FLUID FLOW>NONISOTHERMAL FLOW								
Laminar Flow[2]	—	u, p, T	√	√				
Turbulent flow, Algebraic yPlus[2]	—	$u, p, G,$ $yPlus, T$					√	√
Turbulent flow, L-VEL[2]	—	$u, p, G,$ $uPlus, T$					√	√
Turbulent Flow, k-ε[2]	—	u, p, k, ep, T	√	√				
Turbulent Flow, Low Re k-ε[2]	—	$u, p, k, ep, G,$ T					√	√
HEAT TRANSFER								
Heat Transfer in Solids	ht	T	√	√	√	√		
Heat Transfer in Fluids	ht	T	√	√	√	√		
Heat Transfer in Solids and Fluids	ht	T	√	√	√	√		
HEAT TRANSFER>CONJUGATE HEAT TRANSFER								
Laminar Flow[2]	—	u, p, T	√	√				
Turbulent flow, Algebraic yPlus[2]	—	$u, p, G,$ $yPlus, T$					√	√
Turbulent flow, L-VEL[2]	—	$u, p, G,$ $uPlus, T$					√	√
Turbulent Flow, k-ε[2]	—	u, p, k, ep, T	√	√				
Turbulent Flow, Low Re k-ε[2]	—	$u, p, k, ep, G,$ T					√	√

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

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	STATIONARY	TIME DEPENDENT	THERMAL PERTURBATION, EIGENFREQUENCY	THERMAL PERTURBATION, FREQUENCY DOMAIN	STATIONARY WITH INITIALIZATION	TRANSIENT WITH INITIALIZATION
HEAT TRANSFER>RADIATION								
Heat Transfer with Surface-to-Surface Radiation[2]	—	T, J	√	√	√	√		
Heat Transfer with Radiation in Participating Media[2]	—	T, I (radiative intensity)	√	√	√	√		
Heat Transfer with Radiation in Absorbing-Scattering Media[2]	—	T, I (radiative intensity)	√	√	√	√		
Heat Transfer with Radiative Beam in Absorbing Media[2]	—	T, I (radiative intensity)	√	√	√	√		
Surface-to-Surface Radiation	rad	J	√	√				
Radiation in Participating Media	rpm	I (radiative intensity)	√	√				
Radiation in Absorbing-Scattering Media	rasm	I (radiative intensity)	√	√				
Radiative Beam in Absorbing Media	rbam	I (radiative intensity)	√	√				
HEAT TRANSFER>ELECTROMAGNETIC HEATING								
Joule Heating[2]	—	T, V	√	√		√		
Lumped Thermal System	lts	not applicable	√	√				
HEAT TRANSFER>THIN STRUCURES								
Heat Transfer in Shells	htlsh	T	√	√	√	√		
Heat Transfer in Films	htlsh	T	√	√	√	√		
Heat Transfer in Fractures	htlsh	T	√	√	√	√		
HEAT TRANSFER>HEAT AND MOISTURE TRANSPORT								

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	PRESET STUDIES [1]					
			STATIONARY	TIME DEPENDENT	THERMAL PERTURBATION, EIGENFREQUENCY	THERMAL PERTURBATION, FREQUENCY DOMAIN	STATIONARY WITH INITIALIZATION	TRANSIENT WITH INITIALIZATION
Moist Air[2]	—	T, ϕ	√	√				
Moist Porous Media[2]	—	T, ϕ	√	√				
Building Materials[2]	—	T, ϕ	√	√				
HEAT TRANSFER>HEAT AND MOISTURE TRANSPORT>HEAT AND MOISTURE FLOW								
Laminar Flow[2]	—	u, p, T, ϕ	√	√				
Turbulent flow, Algebraic yPlus[2]	—	$u, p, G,$ $yPlus, T, \phi$					√	√
Turbulent flow, L-VEL[2]	—	$u, p, G,$ $uPlus, T, \phi$					√	√
Turbulent Flow, k-ε[2]	—	$u, p, k, ep, T,$ ϕ	√	√				
Turbulent Flow, Low Re k-ε[2]	—	$u, p, k, ep, G,$ T, ϕ					√	√
Porous Media Flow[2]	—	u, p, T, ϕ	√	√				
Heat Transfer in Porous Media	ht	T	√	√	√	√		
Local Thermal Nonequilibrium	ht	T	√	√	√	√		
Heat Transfer in Packed Beds	ht	T	√	√	√	√		
Bioheat Transfer	ht	T	√	√	√	√		
Thermoelectric Effect[2]	—	T	√	√	√	√		

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

[2] Multiphysics interfaces.

Additional Materials Database

The Heat Transfer Module includes additional material databases:

• The Liquids and Gases material database provides temperature-dependent fluid dynamic and thermal properties (dynamic viscosity, heat capacity at constant pressure, density, and thermal conductivity).

• The Bioheat material database contains materials used with the Bioheat Transfer interface. In addition to thermal properties (heat capacity at constant pressure, density, and thermal conductivity), properties for thermal damage analysis of biological tissues are provided.

• The Building material database contains hygroscopic and thermal properties for materials used with the Moisture Transport interface.

> For detailed information, see Liquids and Gases Materials Library, Bioheat Materials Library, Building Material Library, and 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 Multiphysics and how to access the electronic Documentation and Help content.

The Help window is useful as it is connected to the features in the COMSOL Desktop. 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 model or application includes documentation with the theoretical background and step-by-step instructions to create a model or application. The models and applications are available in COMSOL Multiphysics as MPH files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, 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 model or application and its properties, including options to open it or its associated 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.

COMSOL ACCESS AND TECHNICAL SUPPORT

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 a case number are sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
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 Application Gallery	www.comsol.com/models
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 and Evaluating Results in Layered Materials
- Ambient Properties
- Modeling Heat Transfer by Radiation
- Heat Part Library

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 Heat Transfer in Building Materials
- Theory for Harmonic Heat Transfer
- Theory for Lumped Isothermal Domain
- Theory for Heat Transfer in Thin Structures
- Theory for the Lumped Thermal System Interface
- 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 the section Theory for the Heat Transfer Multiphysics Couplings.

Finally, topics related to specific features or variables are treated in the sections Theory for Thermal Contact, Theory for Heat Transfer in Moist Air, Out-of-Plane Heat Transfer, Convective Heat Transfer Correlations, 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 chapter The Heat Transfer Module Interfaces 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 Shells (htlsh), Radiation in Participating Media (rpm), Radiation in Absorbing-Scattering Media (rasm), Radiative Beam in Absorbing Media (rbam), Surface-to-Surface Radiation (rad), and Moisture Transport (mt)).

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

The sections The Heat Transfer in Porous Media Interface, The Local Thermal Nonequilibrium Interface, and The Heat Transfer in Moist Porous Media Interface discuss modeling heat transfer in porous media.

The particular case of heat transfer in moist air and building materials is considered in the sections The Heat Transfer in Moist Air Interface and The Heat Transfer in Building Materials Interface.

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

The sections The Heat Transfer in Shells Interface, The Heat Transfer in Films Interface, and The Heat Transfer in Fractures Interface describe the physics interfaces which are suitable for solving thermal conduction, convection, and radiation problems in layered materials defined on boundaries.

The section The Lumped Thermal System Interface describes the modeling of heat transfer in a system using a thermal network representation.

The sections The Surface-to-Surface Radiation Interface, The Radiation in Participating Media Interface, The Radiation in Absorbing-Scattering Media Interface, and The Radiative Beam in Absorbing Media Interface discuss the modeling of radiative heat transfer in transparent and participating media.

Finally, the sections The Moisture Transport in Air Interface, The Moisture Transport in Porous Media Interface, and The Moisture Transport in Building Materials Interface describe the modeling of moisture transfer in a porous medium through moisture storage, vapor diffusion and capillary moisture flows; or in air, through convection and diffusion.

THE HEAT TRANSFER FEATURES

The chapter The Heat Transfer Features 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 chapter The Nonisothermal Flow and Conjugate Heat Transfer Interfaces describes the multiphysics versions of both the Nonisothermal 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 section The Heat Transfer with Surface-to-Surface Radiation Interface describes the predefined multiphysics interface used to model heat transfer by conduction, convection, and radiation in a transparent media.

The section The Heat Transfer with Radiation in Participating Media Interface describes the predefined multiphysics interface used to model heat transfer by conduction, convection, and radiation in semitransparent media. The radiative intensity equations are approximated by the Discrete Ordinates Method or the P1 Approximation. When no emission should be considered, see the section The Heat Transfer with Radiation in Absorbing-Scattering Media Interface.

The section The Heat Transfer with Radiative Beam in Absorbing Media Interface describes the predefined multiphysics interface used to model heat transfer by conduction, convection, and radiation in semitransparent media. The Beer-Lambert law is used for the approximation of the radiative intensity.

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

The section The Heat and Moisture Transport Interfaces describes the predefined multiphysics interfaces used to model coupled heat and moisture transport either in building materials, by taking into account heat and moisture storage, latent heat effects, and liquid and convective transport of moisture; or in moist air by convection and diffusion of moisture and heat.

The section The Moisture Flow Interfaces describes the predefined multiphysics interfaces used to model moisture transport in air by laminar flow (in free and porous media) and turbulent flows (in free media).

The section The Heat and Moisture Flow Interfaces describes the predefined multiphysics interfaces used to model heat transfer and moisture transport in air by laminar and turbulent flows, and in porous media by laminar flows.

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

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

Symbols

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
B_i	m	Spectral band i
b	dimensionless	Thermal conductivity supplement
c	mol/m^3	Concentration
c	kg/m^3	Concentration
c_l	mol/m^2	Liquid water concentration on moist surface
$C_{p,b}$	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
$C_{p,A}$	J/(kg·K)	Specific heat capacity at constant pressure, Austenite state
$C_{p,d}$	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
$C_{p,\mathrm{imf}}$	J/(kg·K)	Specific heat capacity at constant pressure of immobile fluid in porous media
$C_{p,\mathrm{imf}i}$	J/(kg·K)	Specific heat capacity at constant pressure of immobile fluid i in porous media
$C_{p,m}$	J/(kg·K)	Specific heat capacity at constant pressure, mixture (moist air)
$C_{p,M}$	J/(kg·K)	Specific heat capacity at constant pressure, Martensite state
$C_{p,s}$	J/(kg·K)	Specific heat capacity at constant pressure, solid phase
$C_{p,si}$	J/(kg·K)	Specific heat capacity at constant pressure of solid material i in porous media
$C_{p,v}$	J/(kg·K)	Specific heat capacity at constant pressure, water vapor

NOTATION	SI UNIT	DESCRIPTION
$C_{p,\,\mathrm{s}}$	J/(kg·K)	Specific heat capacity at constant pressure of thin layer
$C_{p,\,\mathrm{s}i}$	J/(kg·K)	Specific heat capacity at constant pressure of layer i in thin layer
c_{sat}	mol/m^3	Saturation concentration of water vapor
c_{ustr}	mol/m^3	Upstream concentration of water vapor
$C_{p,\,\mathrm{w}}$	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
D	m^2/s	Vapor diffusion coefficient in air
D_{eff}	m^2/s	Vapor diffusion coefficient in moist porous media
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_{pe}	m	Average pellets diameter
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_{T}	m^2/s	Turbulent diffusivity
D_{w}	m^2/s	Moisture diffusivity
d_z	m	Thickness of domain in the out-of-plane direction
$\mathrm{DPT}_{\mathrm{amb}}$	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

NOTATION	SI UNIT	DESCRIPTION
$e_{b,\lambda}(\lambda, T)$	W/m^2	Blackbody spectral emissive power
$e_b(T)$	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
E_p	J/kg	Potential energy
E_u	Pa	Young's modulus, up contact surface
$\mathbf{e}_{\text{tot}}$	W/m^2	Total energy flux
$\mathbf{F}$	N/m^3	Body force vector
$\mathbf{F}$	dimensionless	Deformation gradient tensor
F_{amb}	dimensionless	Ambient view factor
$F_{\text{amb, d}}$	dimensionless	Ambient view factor, downside
$F_{\text{amb, u}}$	dimensionless	Ambient view factor, upside
g	m/s^2	Acceleration of gravity
G_{evap}	kg/m^3	Evaporation source
g_{evap}	kg/(m^2·s)	Evaporation flux
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
$G_{\text{amb, d}}$	W/m^2	Ambient irradiation, downside
$G_{\text{amb, u}}$	W/m^2	Ambient irradiation, upside
G_d	W/m^2	Surface irradiation, downside
G_{ext}	W/m^2	External irradiation
$G_{\text{ext, d}}$	W/m^2	External irradiation, downside
$G_{\text{ext, u}}$	W/m^2	External irradiation, upside
G_m	W/m^2	Mutual surface irradiation
$G_{m, d}$	W/m^2	Mutual surface irradiation, downside
$G_{m, u}$	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

NOTATION	SI UNIT	DESCRIPTION
H	m	Chimney height (heat transfer coefficient)
H	J/kg	Enthalpy
h	W/(m²·K)	Gap conductance (thermal contact)
h	W/(m²·K)	Heat transfer coefficient
H_0	J/kg	Total enthalpy
H_B	Pa	Brinell hardness
h_c	W/(m²·K)	Constriction conductance
H_c	Pa	Microhardness
h_d	W/(m²·K)	Out-of-plane heat transfer coefficient, downside
H_d	J/kg	Enthalpy, downside
H_{ext}	J/kg	External enthalpy
h_g	W/(m²·K)	Parallel-plate gap gas conductance
h_m	m/s	Moisture transfer coefficient
h_r	W/(m²·K)	Radiative conductance
H_{ref}	J/kg	Reference enthalpy
h_{sf}	W/(m²·K)	Interstitial heat transfer coefficient
h_{th}	W/K	Thermal conductance (isothermal domain interface)
h_u	W/(m²·K)	Out-of-plane heat transfer coefficient, upside
H_u	J/kg	Enthalpy, upside
h_z	W/(m²·K)	Out-of-plane heat transfer coefficient, 1D
$I(\Omega)$	W/(m²·sr)	Radiative intensity traveling in direction Ω
$I_b(T)$	W/(m²·sr)	Blackbody radiative intensity
I_{diff}	W/m²	Diffuse irradiance
I_{ext}	W/(m²·sr)	Incident radiative intensity
I_i	W/(m²·sr)	Radiative intensity traveling in ith discrete direction
$\mathbf{i}_s$	dimensionless	Incident radiation direction (external radiation source)
I_s	W/m²	Solar irradiance
$I_{s,amb}$	W/m²	Ambient solar irradiance
$I_{sh,amb}$	W/m²	Clear sky noon diffuse horizontal irradiance
$I_{sn,amb}$	W/m²	Clear sky noon beam normal irradiance
is_x, is_y, is_z	dimensionless	Solar source direction vector components

NOTATION	SI UNIT	DESCRIPTION
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 nonisothermal flow)
k_{a}	W/(m·K)	Dry air thermal conductivity
k_{A}	W/(m·K)	Thermal conductivity of Austenite state
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
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_{imf}	W/(m·K)	Thermal conductivity of immobile fluid in porous media
k_{gap}	W/(m·K)	Parallel-plate gap gas thermal conductivity
$k_{\text{imf}i}$	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_{M}	W/(m·K)	Thermal conductivity of Martensite state
k_{R}	W/(m·K)	Rosseland radiative conductivity
k_{s}	W/(m·K)	Thermal conductivity, solid phase
$k_{\text{s}i}$	W/(m·K)	Thermal conductivity of solid material i in porous media and fractures
k_{s}	W/(m·K)	Thermal conductivity of thin layer
$k_{\text{s}i}$	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	m/s	Evaporation rate factor
K_{Ω}	J	Kinetic energy of a body
L	J/kg	Latent heat

NOTATION	SI UNIT	DESCRIPTION
L	m	Total edge length
L	m	Wall height or plate diameter, distance or length (heat transfer coefficient)
$L_{d,c}$	J/kg	Damage enthalpy change, cryogenic analysis
$L_{d,h}$	J/kg	Damage enthalpy change, hyperthermia analysis
$L_{it,c}$	J/kg	Transformation enthalpy change, overcooling analysis
$L_{it,h}$	J/kg	Transformation enthalpy change, overheating analysis
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)
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
$\mathbf{P}$	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

NOTATION	SI UNIT	DESCRIPTION
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
$\mathrm{Pr_T}$	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
p_{ustr}	Pa	Upstream absolute pressure
p_{v}	Pa	Water vapor partial pressure
$\mathbf{q}$	W/m^2	Conductive heat flux
$\mathbf{q}_{\mathrm{f}}$	W/m^2	Conductive heat flux in fluid phase
$\mathbf{q}_{\mathrm{s}}$	W/m^2	Conductive heat flux in solid phase
Q	W/m^3	Heat source
q_0	W/m^2	Inward heat flux
Q_0	W/m^3	Distributed heat source
$q_{0,\,\mathrm{d}}$	W/m^2	Out-of-plane heat flux, downside
$q_{0,\,\mathrm{s}}$	W/m^2	Source heat flux (external radiation source)
$q_{0,\,\mathrm{u}}$	W/m^2	Out-of-plane heat flux, upside
Q_{b}	W/m^2	Boundary heat source
$Q_{\mathrm{b,\,tot}}$	W/m^2	Total boundary heat source
Q_{e}	W/m^3	Electromagnetic heat source
Q_{evap}	W/m^3	Latent heat source in domains
$\mathbf{q}_{\mathrm{evap}}$	W/m^2	Latent heat source on surfaces
Q_{exch}	W	Exchanged heat source rate

NOTATION	SI UNIT	DESCRIPTION
Q_f	W/m^3	Heat source in fluid phase
q_{geo}	W/kg	Radiogenic heating per mass
Q_{geo}	W/m^3	Geothermal heat source
Q_{Int}	W	Total heat source over interior boundaries
Q_{it}	W/m^3	Irreversible transformation heat source
Q_l	W/m	Line heat source
Q_{met}	W/m^3	Metabolic heat source
Q_p	W	Point heat source
Q_p	W/m^3	Pressure work
Q_r	W/m^3	Radiative heat source term
$\mathbf{q}_r$	W/m^2	Radiative heat flux
$q_{r,\,net}$	W/m^2	Net radiative heat flux
$q_{r,\,out}$	W/m^2	Radiative heat flux striking the wall
q_s	W/(m^3·K)	Production/absorption coefficient
Q_s	W/m^3	Heat source in solid phase
Q_s	W/m^3	Thin layer heat source
q_{sf}	W/(m^3·K)	Interstitial convective heat transfer coefficient
Q_{si}	W/m^3	Layer i heat source
Q_{ted}	W/m^3	Thermoelastic damping
$\mathbf{q}_{tot}$	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
Re$_{inf}$	dimensionless	Reynolds number at infinity
Re$_L$	dimensionless	Reynolds number associated with characteristic length L
Re$_p$	dimensionless	Particle Reynolds number

NOTATION	SI UNIT	DESCRIPTION
r_h	m	Hydraulic radius
r_l	m	Rod radius
r_{pe}	m	Average pellets radius
R_s	J/(kg·K)	Specific gas constant
$R_{t,s}$	K·m^2/W	Surface thermal resistance
R_t	K/W	Thermal resistance
S_b	1/m	Specific surface area
s_d	m	Vapor diffusion equivalent air layer thickness
$\mathbf{S}$	N/m^2	Second Piola-Kirchhoff stress tensor
S	V/K	Seebeck coefficient
Sc_T	dimensionless	Turbulent Schmidt number
$\mathbf{S}_i$	dimensionless	Unit vector of discrete direction in space, i-th component (angular space discretization)
Sp	dimensionless	Sparrow number
T	K	Temperature
T_0	K	Equilibrium temperature
T'	K	Complex amplitude of harmonic perturbation
T^+	dimensionless	Dimensionless temperature
T_{amb}	K	Ambient temperature
$T_{amb, d}$	K	Ambient temperature, downside
$T_{amb, u}$	K	Ambient temperature, upside
T_b	K	Arterial blood temperature
T_d	K	Temperature, downside
$t_{d, c}$	s	Damage time, cryogenic analysis
$T_{d, c}$	K	Damage temperature, cryogenic analysis
$t_{d, h}$	s	Damage time, hyperthermia analysis
$T_{d, h}$	K	Damage temperature, hyperthermia analysis
T_{ext}	K	External temperature
$T_{ext, d}$	K	Out-of-plane external temperature, downside
$T_{ext, u}$	K	Out-of-plane external temperature, upside
$T_{ext, z}$	K	Out-of-plane external temperature, 1D
T_f	K	Temperature, fluid phase

NOTATION	SI UNIT	DESCRIPTION
$t_{\text{it, c}}$		Transformation time, overcooling analysis
$T_{\text{it, c}}$		Transformation temperature, overcooling analysis
$t_{\text{it, h}}$		Transformation time, overheating analysis
$T_{\text{it, h}}$		Transformation temperature, overheating analysis
$T_{\text{n, c}}$	K	Temperature of necrosis, cryogenic
$T_{\text{n, h}}$	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_{ustr}	K	Upstream temperature
T_{w}	K	Wall temperature
$\mathbf{u}$	m/s	Fluid velocity vector
u, v, w	m/s	Fluid velocity vector's components
$\mathbf{u}_{\text{f}}$	m/s	Average linear velocity
$\mathbf{u}_{\text{p}}$	m/s	Porous velocity field
$\mathbf{u}_{\text{trans}}$	m/s	Translational motion velocity vector
v_{amb}	m/s	Wind velocity
V	V	Electric potential
V	m^3	Total domain volume
W	W/m^3	Work source
w	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)
$\mathbf{x}_{\text{s}}$	m	Source location (external radiation source)
X_{v}	dimensionless	Molar fraction of water vapor
x_{vap}	dimensionless	Moisture content
$x_{\text{vap,amb}}$	dimensionless	Ambient moisture content
Y	m	Parallel-plate gap mean separation thickness

NOTATION	SI UNIT	DESCRIPTION
$(\rho C_p)_{\text{eff}}$	J/(m^3·K)	Effective volumetric heat capacity at constant pressure
$\partial\Omega$	-	Geometry domain's boundaries
$\partial\Omega_{\text{ext}}$	-	Geometry domain's exterior boundaries
$\partial\Omega_{\text{int}}$	-	Geometry domain's interior boundaries
$\nabla_{\mathbf{t}}$	-	Tangential gradient operator
α	1/K	Coefficient of thermal expansion tensor in a solid
α	dimensionless	Degree of tissue injury (Arrhenius equation)
α	dimensionless	Irreversible transformation indicator (Arrhenius equation)
α	dimensionless	Parallel-plate gap gas thermal accommodation parameter
α	dimensionless	Surface absorptivity
α	m^2/s	Thermal diffusivity
α	dimensionless	Damaged tissue indicator
α_{necr}	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_p	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
$\gamma_{T\text{eq}}$	Pa/K	Psychrometer constant
ΔR	m	Size of transition zone (deposited beam power)
ΔT	K	Temperature offset for periodic condition
ΔT_0	K	Temperature perturbation
δ	s	Vapor permeability of still air
δ_{p}	s	Vapor permeability
δ_{w}	m	Distance between the computational fluid domain and the wall
ε_{p}	dimensionless	Porosity
ε	dimensionless	Surface emissivity
ε_{d}	dimensionless	Surface emissivity, downside

NOTATION	SI UNIT	DESCRIPTION
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_{imf}	dimensionless	Volume fraction of immobile fluid in porous media
θ_{imfi}	dimensionless	Volume fraction of immobile fluid i in porous media
θ_{it}	dimensionless	Fraction of transformation
θ_g	dimensionless	Volume fraction of mobile fluid in porous media
θ_s	dimensionless	Volume fraction of solid material in porous media and fractures
θ_{si}	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 nonisothermal 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
$\lambda_{tr, h}$	m	Transverse horizontal dispersivity
$\lambda_{tr, v}$	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)
μ_T	Pa·s	Turbulent dynamic viscosity
μ_{Th}	V/K	Thomson coefficient

NOTATION	SI UNIT	DESCRIPTION
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_{imf}	kg/m^3	Density of immobile fluid in porous media
ρ_{geo}	kg/m^3	Geothermal density
ρ_{imfi}	kg/m^3	Density of immobile fluid i in porous media
ρ_l	kg/m^3	Density of thin rod
ρ_g	kg/m^3	Mixture density (moist air)
ρ_s	kg/m^3	Density of solid material
ρ_{si}	kg/m^3	Density of solid material i in porous media
ρ_s	kg/m^3	Density of thin layer
ρ_{si}	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·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_w	dimensionless	Relative humidity
ϕ_{amb}	dimensionless	Ambient relative humidity
ϕ	rad	Tilt angle (heat transfer coefficient)
$\phi_{w, ext}$	dimensionless	External relative humidity
$\phi_{w, d}$	dimensionless	Relative humidity, downside
$\phi_{w, s}$	dimensionless	Surface relative humidity

NOTATION	SI UNIT	DESCRIPTION
$\phi_{w,u}$	dimensionless	Relative humidity, upside
$\phi_{w,ustr}$	dimensionless	Upstream relative humidity
$\Phi_{d \to u}$	kg/s	Mass flow rate, positive direction
$\Phi_{u \to d}$	kg/s	Mass flow rate, negative direction
$\varphi(\Omega', \Omega)$	dimensionless	Scattering phase function
φ_s	rad	Azimuth angle of the Sun
$\varphi_{d \to u}$	kg/($m^2 \cdot$s)	Mass flux, positive direction
$\varphi_{u \to d}$	kg/($m^2 \cdot$s)	Mass flux, negative direction
Ψ	J/kg	Force potential
Ω	dimensionless	Unit vector of a direction in space
Ω	-	Geometry domain
ω	dimensionless	Specific humidity
ω	rad/s	Angular frequency
ω_b	1/s	Blood perfusion rate
ω_i	W/($m^2 \cdot$K)	Discrete incident radiation vector, i-th component
ξ	kg/m^3	Moisture storage capacity
ξ	dimensionless	Martensite state volume fraction

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 and Evaluating Results in Layered Materials
- Evaluating the Bulk Temperature
- Ambient Properties
- Modeling Heat Transfer by Radiation
- Specifying the Temperature at a Flow Inlet
- Heat Part Library

Heat Transfer Variables

In this section:

- Predefined Variables
- Global Variables for Heat and Energy Balance
- Domain Fluxes
- Out-of-Plane Domain Fluxes
- Boundary Fluxes (Heat Transfer Interface)
- 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 heat and energy balance. All the variable names begin with the physics interface name (the prefix). By default the Heat Transfer interface prefix is `ht,` and the Heat Transfer in Shells interface prefix is `htlsh`. 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: PREDEFINED HEAT BALANCE VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
Global variables		
dEiInt	Total Accumulated Heat Rate	Global
ntfluxInt	Total Net Heat Rate	Global
WstrInt	Total Stress Power	Global
heatBalance	Heat Balance	Global
Domain variables		
tflux	Total Heat Flux (Heat Transfer Interface)	Domains

TABLE 3-1: PREDEFINED HEAT BALANCE VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
dflux	Conductive Heat Flux (Heat Transfer Interface)	Domains
cflux	Convective Heat Flux (Heat Transfer Interface)	Domains
turbflux	Turbulent Heat Flux (Heat Transfer Interface)	Domains
tflux_u tflux_d tflux_z	Total Out-of-Plane Heat Flux (Heat Transfer Interface)	Out-of-plane domains (1D and 2D), boundaries
rflux_u rflux_d rflux_z	Radiative Out-of-Plane Heat Flux (Heat Transfer Interface)	Out-of-plane domains (1D and 2D), boundaries
q0_u q0_d q0_z	Out-of-Plane Heat Flux (Heat Transfer Interface)	Out-of-plane domains (1D and 2D)
Boundary variables		
ntflux	Normal Total Heat Flux (Heat Transfer Interface)	Boundaries
ndflux	Normal Conductive Heat Flux (Heat Transfer Interface)	Boundaries
ncflux	Normal Convective Heat Flux (Heat Transfer Interface)	Boundaries
rflux	Radiative Heat Flux (Heat Transfer Interface)	Boundaries
q0	Inward Heat Flux (Heat Transfer Interface)	Boundaries
ntflux_u ntflux_d	Internal normal total heat flux, upside and downside (Internal Boundary Heat Fluxes)	Interior boundaries
ndflux_u ndflux_d	Internal normal conductive heat flux, upside and downside (Internal Boundary Heat Fluxes)	Interior boundaries
ncflux_u ncflux_d	Internal normal convective heat flux, upside and downside (Internal Boundary Heat Fluxes)	Interior boundaries

TABLE 3-2: PREDEFINED ENERGY BALANCE VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
Global variables		
dEi0Int	Total Accumulated Energy Rate	Global
ntefluxInt	Total Net Energy Rate	Global
WInt	Total Work Source	Global
energyBalance	Energy Balance	Global
Domain variables		
teflux	Total Energy Flux (Heat Transfer Interface)	Domains
dflux	Conductive Heat Flux (Heat Transfer Interface)	Domains
thflux	Total Enthalpy Flux (Heat Transfer Interface)	Domains
Boundary variables		
nteflux	Normal Total Energy Flux (Heat Transfer Interface)	Boundaries
ndflux	Normal Conductive Heat Flux (Heat Transfer Interface)	Boundaries
nthflux	Normal Total Enthalpy Flux (Heat Transfer Interface)	Boundaries
q0	Inward Heat Flux (Heat Transfer Interface)	Boundaries
nteflux_u nteflux_d	Internal normal total energy flux, upside and downside (Internal Boundary Energy Fluxes)	Interior boundaries
ndflux_u ndflux_d	Internal normal conductive heat flux, upside and downside (Internal Boundary Heat Fluxes)	Interior boundaries
nthflux_u nthflux_d	Internal normal enthalpy flux, upside and downside (Internal Boundary Energy Fluxes)	Interior boundaries

TABLE 3-3: PREDEFINED HEAT SOURCE VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
Global variables		
QInt	Total Heat Source	Global
Domain variables		

TABLE 3-3: PREDEFINED HEAT SOURCE VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
Qtot	Total heat source (Domain Heat Sources)	Domains
Boundary variables		
Qbtot	Total boundary heat source (Boundary Heat Sources)	Boundaries
Qltot	Total line heat source (Line and Point Heat Sources)	Edges, Points (2D, 2D axial symmetry)
Qptot	Point heat source (Line and Point Heat Sources)	Points

TABLE 3-4: OTHER PREDEFINED HEAT TRANSFER VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
Global variables		
Tref	Reference temperature	Global
Domain variables		
alpha	Damage indicator (see Theory for Bioheat Transfer)	Domains
alphanecr	Instant tissue necrosis indicator (see Theory for Bioheat Transfer)	Domains
theta_d	Fraction of damage (see Theory for Bioheat Transfer)	Domains
theta_d_sm	Fraction of total damage (see Theory for Bioheat Transfer)	Domains
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
Boundary variables		
h	Heat transfer coefficient (see Convective Heat Transfer Correlations)	Exterior boundaries

> 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 for Heat and Energy Balance

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

$$\text{dEiInt} = \frac{d}{dt}\int_\Omega \rho E_i d\omega$$

TOTAL NET HEAT RATE

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

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

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:

$$\text{QInt} = \int_\Omega Q d\omega + \int_{\partial\Omega_{int}} Q_b d\omega + \int_{\partial\Omega_{int}} Q_r d\omega$$

TOTAL STRESS POWER

The total stress power, `WstrInt`, corresponds to the work lost by a fluid by
degradation of energy. These works are transmitted to the system through pressure
work and viscous dissipation:

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

TOTAL ACCUMULATED ENERGY RATE

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

$$\text{dEi0Int} \ = \ \frac{d}{dt} \int_\Omega \rho E_{i0} d\omega$$

where the total internal energy, E_{i0}, is defined as

$$E_{i0} \ = \ E_i + \frac{\mathbf{u} \cdot \mathbf{u}}{2}$$

TOTAL NET ENERGY RATE

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

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

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

TOTAL WORK SOURCE

The total work source, `WInt`, accounts for volume forces and boundary stresses as follows:

$$\text{WInt} \ = \ \int_\Omega (\rho \mathbf{g} + \mathbf{F}) \cdot \mathbf{u} d\omega + \int_{\partial\Omega_{ext}} \mathbf{K} \cdot \mathbf{u} d\sigma + \int_{\partial\Omega_{int}} \mathbf{K} \cdot \mathbf{u}|_{\partial\Omega_{int}} d\sigma$$

where $\mathbf{K}$ is the viscous force vector defined as

$$\mathbf{K} \ = \ \left[\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} \right] \cdot \mathbf{n}$$

and $\mathbf{K} \cdot \mathbf{u}|_{\partial\Omega_{int}}$ is the jump of viscous stress at interior boundaries.

HEAT BALANCE

According to Equation 4-200, the following equality between COMSOL Multiphysics variables holds:

```
dEiInt + ntfluxInt = QInt - WstrInt
```

and the heat balance variable, `heatBalance`, is defined as

```
heatBalance = dEiInt + ntfluxInt + WstrInt - QInt
```

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 - WstrInt
```

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 `WstrInt`, the losses that heat up the system are negative and the gains that cool down the system are positive.

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:

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

📑 Depending on the radiation discretization method, the contribution to the heat balance is handled differently. For the definition of `ntfluxInt`, the **Optically Thick Participating Medium** subnode uses the Rosseland approximation and defines $\mathbf{q_r}$, the radiative flux, as an extra contribution to the conductive heat flux. The **P1 approximation** and **Discrete ordinates method**, however, include the radiative source $\nabla \cdot \mathbf{q_r}$ to Q on the domain, in the variable `QInt`.

📑 The heat balance can be evaluated assuming constant material properties using the following variables: `heatBalance_cst`, `dEiInt_cst`, `ntfluxInt_cst`, `WstrInt_cst`, `QInt_cst`. The evaluation of these variables is faster and accurate for constant material properties or small temperature gradient.

According to Equation 4-201, the following equality between COMSOL Multiphysics predefined variables holds:

```
dEiOInt + ntefluxInt = QInt + WInt
```

and the energy balance variable, `energyBalance`, is defined as

```
energyBalance = dEiOInt + ntefluxInt - WInt - QInt
```

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

```
ntefluxInt = QInt + WInt
```

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 `WstrInt` for pressure work and viscous dissipation in fluids. Hence, energy is the conserved quantity, not heat.

📄 | The heat and energy balance variables such as `ht.ntfluxInt` or `ht.ntefluxInt` do not take into account the flux contributions on thin structures. For example, if a **Thin Layer** node is applied on a boundary, the heat flux on this boundary is not included in `ht.ntfluxInt`. It is added instead to the variable `ht.sls1.ntfluxInt`, defined locally on the thin structure. Here, `sls1` is the tag of the **Thin Layer** node.

📄 | The energy balance can be evaluated assuming constant material properties using the following variables: `energyBalance_cst`, `dEiOInt_cst`, `ntefluxInt_cst`, `WInt_cst`, `QInt_cst`. The evaluation of these variables is faster and accurate for constant material properties or small temperature gradient.

Domain Fluxes

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

TOTAL HEAT FLUX (HEAT TRANSFER INTERFACE)

In domains, the total heat flux, tflux, is the sum of the conductive and convective heat fluxes. For accuracy reasons the radiative heat flux is not included.

> 💡 See Radiative Heat Flux (Heat Transfer Interface) to evaluate the radiative heat flux.

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

$$tflux = dflux$$

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

$$tflux = cflux + dflux$$

CONDUCTIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

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

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

For heat transfer in fluids with turbulent flow, $k_{eff} = k + k_T$, 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:

$$dflux = -(k_{eff}\nabla T + L_v \delta_p \nabla(\phi_w p_{sat}))$$

When considering thermoelectric effect, the thermoelectric heat flux is also included in the diffusive flux, see Equation 4-150.

TURBULENT HEAT FLUX (HEAT TRANSFER INTERFACE)

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

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

CONVECTIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

$$\text{cflux} = \rho \mathbf{u} E_\text{i}$$

The internal energy, E_i, is defined as:

- $E_\text{i} = H$ for solid domains
- $E_\text{i} = H - p/\rho$ for fluid domains

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

The convective heat flux may be oriented in the opposite direction of the velocity field's direction, when the internal energy, E_i, has a negative value. This happens when the sensible enthalpy (variation from reference enthalpy H_ref) is negative. H_ref is set to 0 J/kg at p_ref (1 atm), c_ref (0 mol/m^3), and T_ref (293.15 K) in COMSOL Multiphysics. See Thermodynamic Description of Heat Transfer for details.

RADIATIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

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 $Q_\text{r} = \nabla \cdot q_\text{r}$.

TOTAL ENTHALPY FLUX (HEAT TRANSFER INTERFACE)

The total enthalpy flux variable, `thflux`, is defined as:

$$\text{thflux} = \rho \mathbf{u} H_0$$

where the total enthalpy, H_0, is defined as

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

TOTAL ENERGY FLUX (HEAT TRANSFER INTERFACE)

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

$$\text{teflux} = \text{thflux} + \text{dflux} + \tau \mathbf{u}$$

Out-of-Plane Domain Fluxes

RADIATIVE OUT-OF-PLANE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

- In 2D:

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

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

- In 1D:

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

OUT-OF-PLANE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

- In 2D:

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

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

- In 1D:

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

TOTAL OUT-OF-PLANE HEAT FLUX (HEAT TRANSFER INTERFACE)

The total out-of-plane heat flux, `tflux`, is the sum of the radiative and convective heat fluxes:

- In 2D:

$$\text{upside: tflux\_u} = \text{q0\_u} + \text{rflux\_u}$$

$$\text{downside: tflux\_d} = \text{q0\_d} + \text{rflux\_d}$$

- In 1D:

$$\text{tflux\_z} = \text{q0\_z} + \text{rflux\_z}$$

Boundary Fluxes (Heat Transfer Interface)

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

NORMAL TOTAL HEAT FLUX (HEAT TRANSFER INTERFACE)

The variable `ntflux` is defined as:

$$ntflux = ndflux + ncflux$$

NORMAL CONDUCTIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

- `ndflux = -dflux_spatial`(T) if the adjacent domain is on the downside,
- `ndflux = -uflux_spatial`(T) if the adjacent domain is on the upside,

and, on interior boundaries, as:

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

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

NORMAL CONVECTIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

The variable `ncflux` is defined as:

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

NORMAL TOTAL ENERGY FLUX (HEAT TRANSFER INTERFACE)

The variable `nteflux` is defined as:

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

NORMAL TOTAL ENTHALPY FLUX (HEAT TRANSFER INTERFACE)

The variable `nthflux` is defined as:

$$nthflux = \text{mean}(thflux) \cdot \mathbf{n}$$

INWARD HEAT FLUX (HEAT TRANSFER INTERFACE)

On external boundaries, the inward heat flux, q0, is a scalar quantity defined either in a general way or as a convective heat flux as follows:

$$q0 = h(T_{ext} - T)$$

where T_{ext} is the external temperature defined in the Heat Flux feature.

RADIATIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

$$rflux = \varepsilon(G - e_b(T)) + q_{r, net}$$

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 (HEAT TRANSFER INTERFACE)

The variable ndflux_u is defined as:

$$ndflux\_u = uflux\_spatial(T)$$

INTERNAL NORMAL CONDUCTIVE HEAT FLUX, DOWNSIDE (HEAT TRANSFER INTERFACE)

The variable ndflux_d is defined as:

$$ndflux\_d = dflux\_spatial(T)$$

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

INTERNAL NORMAL CONVECTIVE HEAT FLUX, UPSIDE (HEAT TRANSFER INTERFACE)

The variable ncflux_u is defined as:

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

INTERNAL NORMAL CONVECTIVE HEAT FLUX, DOWNSIDE (HEAT TRANSFER INTERFACE)

The variable ncflux_d is defined as:

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

INTERNAL NORMAL TOTAL HEAT FLUX, UPSIDE (HEAT TRANSFER INTERFACE)

The variable ntflux_u is defined as:

$$ntflux\_u \,=\, ndflux\_u + ncflux\_u$$

INTERNAL NORMAL TOTAL HEAT FLUX, DOWNSIDE (HEAT TRANSFER INTERFACE)

The variable ntflux_d is defined as:

$$ntflux\_d \,=\, ndflux\_d + ncflux\_d$$

Internal Boundary Energy Fluxes

INTERNAL NORMAL TOTAL ENERGY FLUX, UPSIDE (HEAT TRANSFER INTERFACE)

The variable nteflux_u is defined as:

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

INTERNAL NORMAL TOTAL ENERGY FLUX, DOWNSIDE (HEAT TRANSFER INTERFACE)

The variable nteflux_d is defined as:

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

INTERNAL NORMAL ENTHALPY FLUX, UPSIDE (HEAT TRANSFER INTERFACE)

The variable nthflux_u is defined as:

$$nthflux\_u \,=\, up(thflux) \cdot \mathbf{un}$$

The variable `nthflux_d` is defined as:

$$\text{nthflux\_d} = \text{down}(\text{thflux}) \cdot \mathbf{dn}$$

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 Heating (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_{dmg}, which is the cooling source added by the Irreversible Transformation feature.

- Q_r, which is the radiative heat source added by the Heat Transfer with Radiation in Participating Media, Heat Transfer with Radiation in Absorbing-Scattering Media, and Heat Transfer with Radiative Beam in Absorbing Media multiphysics features.

- Q_{geo}, which is the geothermal heat source added by the Geothermal Heating feature.

- Q_{evap}, which is the latent heat source added by the Heat and Moisture coupling node.

Boundary Heat Sources

The sum of the boundary heat sources added by different boundary conditions is available in the variable, $Q_{b,tot}$ (SI unit: 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 Electromagnetic Heating 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 Heat Source (Thin Layer, Thin Film, Fracture).

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:

$$p_{sat}(\text{T\_dp}) = p_v = \text{phi} \cdot \text{psat}$$

where `phi` is the Relative Humidity variable. See Saturation State for the definition of saturation pressure p_{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:

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

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

$$\gamma_{T\mathrm{eq}} = \frac{M_a p C_{p,a}}{M_v L_v}$$

where p is the total pressure, $C_{p,a}$ 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.
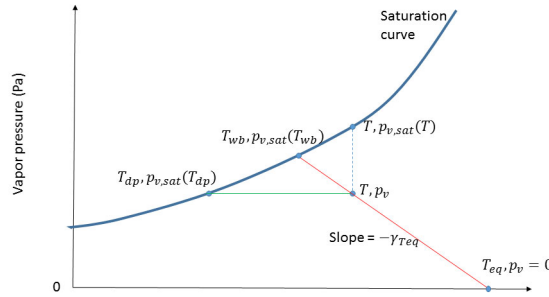
Figure 3-1: Relation between dry bulb, dew point, equivalent, and wet bulb temperatures.

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:

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

The psychrometer constant is again evaluated at temperature T.

SATURATION PRESSURE

The variable `psat` is defined by:

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

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

RELATIVE HUMIDITY

The variable `phi` is defined by:

$$phi = \frac{p_v}{p_{sat}(T)}$$

LATENT HEAT OF EVAPORATION

The variable `Lv` is defined by:

$$Lv = lv(T)$$

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 3-5: MOISTURE TRANSPORT PREDEFINED VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
dwcInt	Total accumulated moisture rate	Global
ntfluxInt	Total net moisture rate	Global
GInt	Total moisture source	Global
massBalance	Mass balance	Global
phi	Relative humidity	Domains
cv	Vapor concentration	Domains
cl	Liquid water concentration	Domains
cw	Total moisture concentration	Domains
ca	Dry air concentration	Domains
csat	Saturation concentration	Domains
psat	Saturation pressure of water vapor	Domains
pv	Partial pressure of water vapor	Domains
wc_l	Liquid water content	Domains
wc_v	Vapor content	Domains
wcVar	Moisture content	Domains
omega_v	Vapor mass fraction	Domains
omega_moist	Moisture mass fraction	Domains
xvap	Humidity ratio	Domains
Lv	Latent heat of evaporation	Domains
Xa	Molar fraction of dry air	Domains
Xv	Molar fraction of water vapor	Domains
rhoa	Density of dry air	Domains
rhov	Density of water vapor	Domains

TABLE 3-5: MOISTURE TRANSPORT PREDEFINED VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
rho_moist	Density of moist air	Domains
mua	Viscosity of dry air	Domains
muv	Viscosity of water vapor	Domains
mu_moist	Viscosity of moist air	Domains
cflux	Convective moisture flux	Domains
dflux	Diffusive moisture flux	Domains
tflux	Total moisture flux	Domains
tfluxMag	Total moisture flux magnitude	Domains
Gtot	Total moisture source	Domains
dwc	Total accumulated moisture rate density	Domains
h_m	Moisture transfer coefficient	Boundaries
ncflux	Normal convective moisture flux	Boundaries
ndflux	Normal diffusive moisture flux	Boundaries
ntflux	Normal total moisture flux	Boundaries
g_evap	Evaporation flux	Boundaries
q_evap	Latent heat source	Boundaries
q_evaptot	Total latent heat source on surfaces	Boundaries
gtot	Total moisture flux	Boundaries
uStefan_u	Stefan velocity, upside	Boundaries
uStefan_d	Stefan velocity, downside	Boundaries
ncflux_u	Normal convective moisture flux, upside	Interior boundaries
ncflux_d	Normal convective moisture flux, downside	Interior boundaries
ndflux_u	Normal diffusive moisture flux, upside	Interior boundaries
ndflux_d	Normal diffusive moisture flux, downside	Interior boundaries
ntflux_u	Normal total moisture flux, upside	Interior boundaries
ntflux_d	Normal total moisture flux, downside	Interior boundaries

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`:

$$pv = phi \cdot psat$$

SATURATION CONCENTRATION OF WATER VAPOR

The saturation concentration `csat` is obtained from the saturation pressure `psat` and temperature T by:

$$csat = \frac{psat}{RT}$$

where R is the universal gas constant.

HUMIDITY RATIO

The humidity ratio `xvap` is obtained from the vapor and air concentrations c_v and c_a, the absolute pressure p_A and temperature T by:

$$x_{vap} = \frac{c_v}{c_a} \cdot \frac{M_v}{M_a}$$

where R is the universal gas constant, M_v is the water vapor molar mass, and M_a is the dry air molar mass.

The vapor and dry air concentrations satisfy:

$$c_a = \frac{p_A}{RT} - c_v$$

VAPOR MASS FRACTION

The vapor mass fraction `omega_v` is obtained by:

$$\omega_v = \frac{c_v M_v}{c_a M_a + c_v M_v}$$

LIQUID, VAPOR, AND MOISTURE CONCENTRATIONS

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

$$cw = cv + cl$$

Note that `cl=0` in moist air domains.

CONCENTRATIONS AND CONTENTS

The liquid water and vapor contents `wc_l` and `wc_v` are obtained by multiplying the concentrations `cl` and `cv` by the molar mass of water `M_v`:

$$wc\_l = \overline{M\_v} \cdot cl$$

$$wc\_v = M\_v \cdot cv$$

MOISTURE CONTENT

The moisture content `wcVar` is expressed in different ways depending on the domain type:

- In moist air:

$$w(\phi_w) = \rho_g \omega_v$$

- In hygroscopic porous media:

$$w(\phi_w) = \varepsilon_p(\rho_l s_l + \rho_g \omega_v(1 - s_l))$$

- In building materials, it is given by the moisture storage function, which may be given as a material property.

LATENT HEAT OF EVAPORATION

The variable `Lv` is defined by:

$$Lv = lv(T)$$

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

Domain Moisture Fluxes

CONVECTIVE MOISTURE FLUX

The definition of the variable `cflux` depends on the domain model and on the species formulation:

- In moist air domains, with the diluted species formulation:

$$\text{cflux} = M_v \mathbf{u}_g c_v$$

- In moist air domains, with the concentrated species formulation:

$$\text{cflux} = \rho_g \mathbf{u}_g \omega_v$$

- In hygroscopic porous media domains:

$$\text{cflux} = \rho_g \mathbf{u}_g \omega_v + \rho_l \mathbf{u}_l$$

- In building materials domains:

$$\text{cflux} = -\delta_p \phi_w \nabla(p_{\text{sat}}(T))$$

In building materials domains, the definition of `cflux` does not correspond to convection in the physical sense, but rather to vapor diffusion. However, this term is numerically handled like a convection flux, and is therefore included in the variable `cflux`.

DIFFUSIVE MOISTURE FLUX

The definition of the variable `dflux` depends on the domain model and on the species formulation:

- In moist air domains, with the diluted species formulation:

$$\text{dflux} = -M_v D \nabla c_v$$

- In moist air domains, with the concentrated species formulation:

$$\text{dflux} = -\rho_g D \nabla \omega_v$$

- In hygroscopic porous media domains:

$$\text{dflux} = \mathbf{g}_{lc} - \rho_g D_{\text{eff}} \nabla \omega_v$$

- In building materials domains:

$$\text{dflux} = -\xi D_w \nabla \phi_w - \delta_p p_{\text{sat}}(T) \nabla \phi_w$$

In hygroscopic porous media and building materials domains, the first term of the definition of `dflux` corresponds to liquid capillary flux. This term is numerically handled like a diffusion flux, and is therefore included in the variable `dflux`.

TOTAL MOISTURE FLUX

The variable `tflux` is defined as:

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

TOTAL MOISTURE FLUX MAGNITUDE

The variable `tfluxMag` is defined as:

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

Boundary Moisture Fluxes

NORMAL CONVECTIVE MOISTURE FLUX

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

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

INTERNAL NORMAL CONVECTIVE MOISTURE FLUX, UPSIDE

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

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

INTERNAL NORMAL CONVECTIVE MOISTURE FLUX, DOWNSIDE

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

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

NORMAL DIFFUSIVE MOISTURE FLUX

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

$$\text{ndflux} = \text{mean}(\text{dflux}) \cdot \mathbf{n}$$

INTERNAL NORMAL DIFFUSIVE MOISTURE FLUX, UPSIDE

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

$$\text{ndflux\_u} = \text{up}(\text{dflux}) \cdot \mathbf{un}$$

INTERNAL NORMAL DIFFUSIVE MOISTURE FLUX, DOWNSIDE

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

$$\text{ndflux\_d} = \text{down}(\text{dflux}) \cdot \mathbf{dn}$$

NORMAL TOTAL MOISTURE FLUX

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

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

INTERNAL NORMAL TOTAL MOISTURE FLUX, UPSIDE

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

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

INTERNAL NORMAL TOTAL MOISTURE FLUX, DOWNSIDE

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

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

TOTAL MOISTURE FLUX

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

Domain Moisture Source

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

Mass Balance

The mass balance variable, `massBalance`, is defined as

```
massBalance = dwcInt + ntfluxInt - GInt
```

where:

- the variation of total moisture content per unit time in the domain, `dwcInt`, is:

$$\text{dwcInt} = \frac{d}{dt}\int_{\Omega} \text{w}(\phi_{\text{w}})d\omega$$

- the total net moisture rate, `ntfluxInt`, is the integral of the total moisture flux over all external boundaries:

$$\text{ntfluxInt} = \int_{\partial\Omega_{\text{ext}}} \text{tflux} \cdot \mathbf{n}d\sigma$$

- the total moisture source, `GInt`, is the integral of the moisture sources over the domains:

$$\text{GInt} = \int_{\Omega} \text{Gtot}d\omega$$

Using the Boundary Conditions for the Heat Transfer Interfaces

In this section:

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: 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: 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 $q_0 = h \cdot (T_{ext} - T)$, 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.

Q | 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 (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)); others cannot (for example, Heat Flux and Thermal Insulation).

Several categories of boundary condition exist in heat transfer. Table 3-6 gives the overriding rules for these groups.

1 Temperature, Open Boundary, Inflow, Outflow, Phase Change Interface
2 Thermal Insulation, Symmetry (Heat Transfer Interface), Periodic Condition (Heat Transfer Interface)
3 Heat Flux
4 Boundary Heat Source, Line Heat Source on Axis, Deposited Beam Power
5 Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface), Thermal Contact, Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface), and Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
6 Isothermal Domain Interface

TABLE 3-6: OVERRIDING RULES FOR HEAT TRANSFER BOUNDARY CONDITIONS

A\B	1	2	3	4	5	6
1-Temperature	X	X			X	
2-Thermal Insulation	X	X				
3-Heat Flux	X	X				
4-Boundary heat source						
5-Thin Layer	X				X	
6-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 3-6 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

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 3-7: **Override and Contribution** SECTIONS.

HEAT FLUX	SYMMETRY
Override and Contribution Overridden by: Symmetry 1 {sym1} Overrides: Contributes with:	**Override and Contribution** Overridden by: Overrides: Thermal Insulation 1 {ins1} Heat Flux 1 {hf1} Contributes with:

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

Handling Frames in Heat Transfer

In this section:

- Heat Transfer Analysis with Moving Frames
- Material Density in Features Defined in the Material Frame

Heat Transfer Analysis with Moving Frames

This section discusses heat transfer analysis with moving frames, when spatial and material frames do not coincide. In this case, the heat transfer interfaces account for deformation effects on heat transfer properties.

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

In the heat transfer interfaces, the entire physics (equations and variables) are defined in the spatial frame. When a moving mesh is detected, the user inputs for certain features are defined in the material frame and are converted so that all the corresponding variables contain the value in 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 3-8, Table 3-9, and Table 3-10:

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

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

Material/(Spatial):   For these physics nodes, select from the **Material type** list to decide if the inputs are defined in the material or spatial frame. The default definition frame is the material frame, which corresponds to **Solid** in the **Material type** list.

(Material)/Spatial: For these physics nodes, select from the **Material type** list to decide if the inputs are defined in the material or spatial frame. The default definition frame is the spatial frame, which corresponds to **Nonsolid** in the **Material type** list.

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

DEFINITION FRAME OF DOMAIN NODES

TABLE 3-8: DOMAIN PHYSICS NODES FOR FRAMES

NODE NAME	DEFINITION FRAME
Absorbing Medium	Spatial
Absorbing-Scattering Medium	Spatial
Bioheat	Material
Biological Tissue	Material
Building Material	Material
Cross Section	Spatial
Thickness	Spatial
Fluid	Spatial
Geothermal Heating	Material
Heat Source	Material/(Spatial)
Immobile Fluids	Spatial
Infinite Elements	Spatial
Initial Values	Spatial
Irreversible Transformation	Spatial
Isothermal Domain	Spatial
Moist Air	Spatial
Moist Porous Medium	Material (Solid part) Spatial (Fluid part)
Opacity	N/A
Optically Thick Participating Medium	Spatial
Out-of-Plane Heat Flux	Spatial
Out-of-Plane Radiation	Spatial
Participating Medium	Spatial
Phase Change Material	Spatial
Porous Medium	Material (Solid part) Spatial (Fluid part)
Pressure Work	Spatial
Radiative Source	Material/(Spatial)
Shape Memory Alloy	Material
Solid	Material

TABLE 3-8: DOMAIN PHYSICS NODES FOR FRAMES

NODE NAME	DEFINITION FRAME
Thermal Dispersion	Spatial
Thermoelastic Damping	Spatial
Translational Motion	Material
Viscous Dissipation	Spatial

DEFINITION FRAME OF BOUNDARY NODES

TABLE 3-9: BOUNDARY PHYSICS NODES FOR FRAMES

NODE NAME	DEFINITION FRAME
Boundary Heat Source	Material/(Spatial)
Continuity	Spatial
Continuity on Interior Boundary	Spatial
Deposited Beam Power	Spatial
Diffuse Mirror	Spatial
Diffuse Surface	Spatial
External Temperature	Spatial
Fracture	Material (Solid part) Spatial (Fluid part)
Heat Flux	(Material)/Spatial
Heat Source	Material/(Spatial)
Incident Intensity	Spatial
Inflow	Spatial
Irreversible Transformation	Material
Isothermal Domain Interface	Spatial
Lumped System Connector	Spatial
Opaque Surface	Spatial
Open Boundary	Spatial
Outflow	N/A
Periodic Condition	Spatial
Prescribed Radiosity	Spatial
Radiation Group	N/A
Semitransparent Surface	Spatial
Surface-to-Ambient Radiation	Spatial
Symmetry	N/A
Temperature	Spatial
Thermal Contact	Material
Thermal Damage	Material
Thermal Insulation	N/A

TABLE 3-9: BOUNDARY PHYSICS NODES FOR FRAMES

NODE NAME	DEFINITION FRAME
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 3-10: EDGE AND POINT NODES FOR FRAMES

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

Material Density in Features Defined in the Material Frame

In a model the features defined in the material frame with an input field for the density expect the density in the material frame, defined for the reference geometry. Hence this density is constant in most cases. A non constant density in the material frame means that there is addition or removal of matter.

As a consequence, when the material is defined from a material library and is temperature dependent, a constant reference temperature, the volume reference temperature, is used for the definition of the density in the **Material Contents** table.

The volume reference temperature defines the density in the reference geometry that should match with the geometry in the material frame. It is a model input of all the features defined in the material frame with an input field for the density. Following Table 3-8, Table 3-9, and Table 3-10, this concerns the following features: Solid, Porous Medium, Biological Tissue, Building Material, Shape Memory Alloy, Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface), Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface), and Thin Rod.

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 More Options** button (👁) and select **Stabilization** in the **Show More Options** dialog box.

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.

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
- Selection Information

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, the **Nonisothermal Flow** multiphysics coupling node facilitates the coupling between heat transfer and fluid flow interfaces.

All these options make it possible to build a coupling in different ways. Even if the use of the predefined multiphysics coupling interfaces — **Nonisothermal 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
- Adding Nonisothermal Flow Coupling in an Existing Model
- Nonisothermal 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-ε Interface, and The Turbulent Flow, Low Re k-ε 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 define 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. Pressure work and viscous dissipation are not handled. 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.

Adding Nonisothermal Flow Coupling in an Existing Model

The **Nonisothermal Flow** multiphysics feature can be added to a model containing a single-phase flow and a heat transfer interface, to realize the bidirectional coupling between the two interfaces.

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 **Nonisothermal Flow** multiphysics feature realizes the bidirectional coupling. It also redefines the consistent stabilization so that the multiphysics coupling effects are accounted for in the numerical stabilization.

With the Heat Transfer Module or the CFD Module, 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. It allows to include work done by pressure changes and viscous dissipation, and Boussinesq approximation is supported. Finally some physics features are updated when the **Nonisothermal 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.

Nonisothermal Flow and Conjugate Heat Transfer Multiphysics Interfaces

These Nonisothermal Flow and Conjugate Heat Transfer Multiphysics Interfaces 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 **Nonisothermal 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 wall. The actual definition of `ht.Tvar` depends on the model configuration.

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_u`: upside wall temperature defined by a **Wall** or an **Interior Wall** feature with turbulence only if a **Fluid** feature is defined on the upside of the wall.

- `TWall_d`: downside wall temperature defined by a **Wall** or an **Interior Wall** feature with turbulence only if a **Fluid** feature is defined on the downside of the wall.

- `Tu`: temperature on the upside of the boundary.

- `Td`: temperature on the downside of the boundary.

- `TExtFace`: external temperature of an external boundary defined by a thermally thick boundary condition.

- `TuWF`: temperature of the fluid in the turbulent boundary layer near the wall only if the **Fluid** feature is defined on the upside of the **Wall** feature.

- `TdWF`: temperature of the fluid in the turbulent boundary layer near the wall only if the **Fluid** feature is defined on the downside of the **Wall** feature.

The values of these variables depend on the selections where they are defined. Here are some illustrated cases:

- Intersection Between a Wall Boundary Feature Selection and Interface Exterior Boundaries
- Interface Between a Fluid Domain Feature and a Solid Domain Feature Where a Wall Boundary Feature Is Active
- Interface Between 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, and `TuWF = Tu`, `TdWF = Td`.
- Turbulence models: wall functions detected by the Single Phase Flow physics interface. `TuWF` and `TdWF` have different definitions and a flux `qwf` is applied on the wall boundaries. See Temperature Condition for Automatic Wall Treatment and Wall functions in the *CFD Module User's Manual* for details.

The following sections summarize the definitions of the temperature variables for the abovementioned configurations.

INTERSECTION BETWEEN A WALL BOUNDARY FEATURE SELECTION AND INTERFACE EXTERIOR BOUNDARIES

The following figure shows a configuration with a fluid domain, a thermally thick boundary condition on the left boundary, and a **Wall** feature on the right boundary. This example uses wall functions.
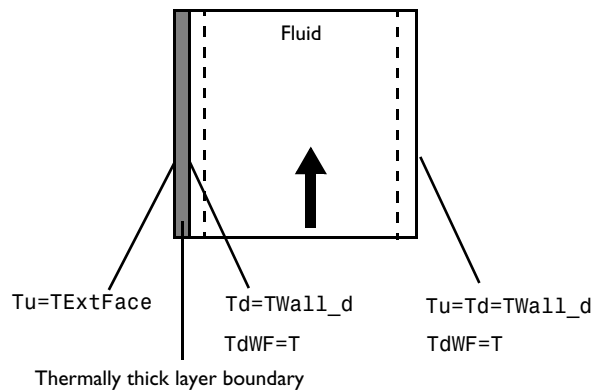
Fluid

Tu=TExtFace   Td=TWall_d   Tu=Td=TWall_d
                 TdWF=T   TdWF=T

Thermally thick layer boundary

INTERFACE BETWEEN A FLUID DOMAIN FEATURE AND A SOLID DOMAIN FEATURE WHERE A WALL BOUNDARY FEATURE IS ACTIVE

Solid in the Domain Downside the Wall

The following figures show configurations with a solid as downside domain and a fluid as upside domain with either a thermally thick or thermally thin boundary condition in between. This example uses wall functions.


```
Td=down(T)        Tu=TWall_u        Tu=Td=TWall_d
                  TuWF=up(T)        TdWF=T
```

Thermally thick layer boundary


```
Tu=Td=down(T)
                              Tu=Td=TWall_d
```

Thermally thin layer boundary
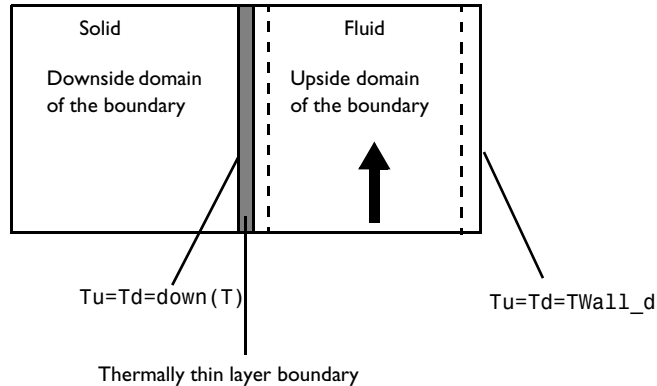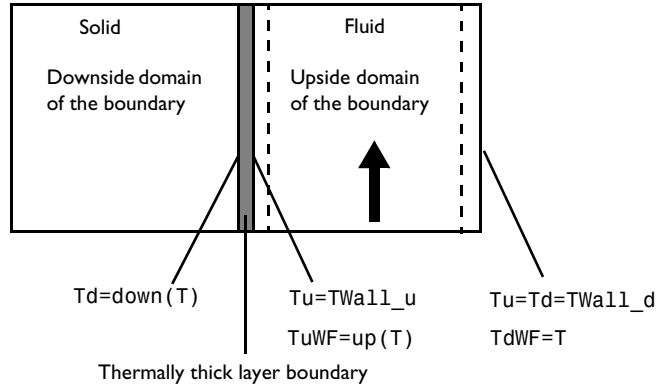
Solid in the Domain Upside the Wall

The following figures show configurations with a fluid as downside domain and a solid as upside domain with either a thermally thick or thermally thin boundary condition in between. This example uses wall functions.


```
Tu=Td=Twall_d
```
```
Td=TWall_d
```
```
Tu=up(T)
```
```
TdWF=down(T)
```

Thermally thick layer boundary


```
Tu=Td=Twall_d
```
```
Tu=Td=up(T)
```

Thermally thin layer boundary
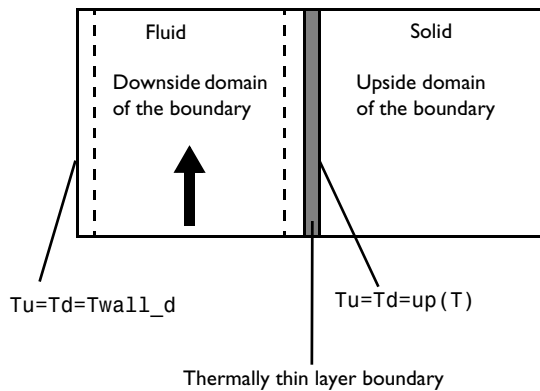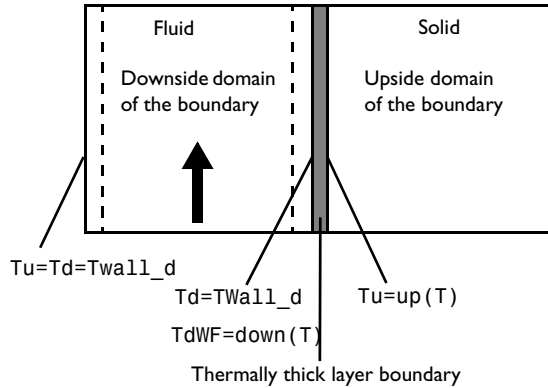
INTERFACE BETWEEN TWO FLUID DOMAIN FEATURES WHERE AN INTERIOR WALL BOUNDARY FEATURE IS ACTIVE

The following figure shows a configuration with two fluid domains with a thermally thick boundary condition and an **Interior Wall** feature in between. This example uses wall functions.
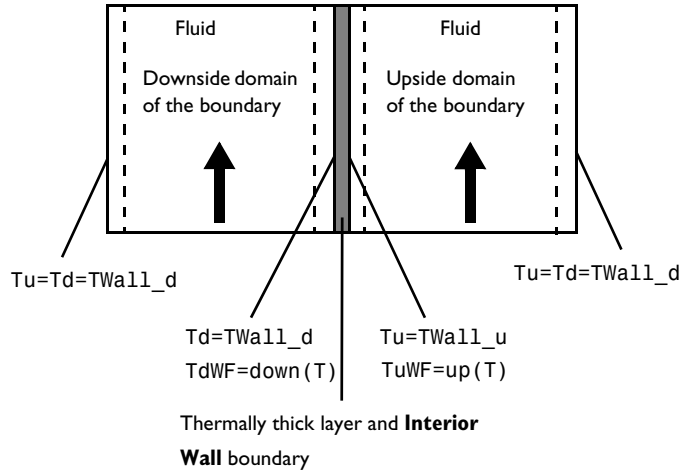
Solver Settings

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*

In this section:

- Linearity Property of the Temperature Equation
- Linear Solver
- Nonlinear Solver
- Time-Dependent Study Step
- Guidelines for Solving Surface-to-Surface Radiation Problems
- Guidelines for Solving Multiphysics Problems

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 $-\mathbf{n} \cdot \mathbf{q} = h(T_{ext} - T)$ keeps the linearity of the problem when the heat transfer coefficient, h, is constant.
  - A radiative condition of type $-\mathbf{n} \cdot \mathbf{q} = \varepsilon\sigma(T_{amb}^4 - T^4)$ is strongly nonlinear.

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

Linear Solver

DEFAULT LINEAR SETTINGS FOR HEAT TRANSFER AND MOISTURE TRANSPORT 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 is the presmoother and postsmoother. This solver is memory effective and fast for large models.

When the model contains settings that lead to a system matrix with 0 on the diagonal (for example, Lagrange multipliers for weak constraints), SOR cannot be used and 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 NONLINEAR 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 use a fully coupled nonlinear solver attribute by default. The **Jacobian update** is set to **On every iteration** for **Stationary** studies, and to **On first iteration** for **Time-Dependent** studies. A Newton nonlinear method is set by default with

- Automatic damping factor computation for stationary studies
- Constant damping factor for time-dependent studies, with a default damping factor set to 0.9.

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 168 for a single wavelength), 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. When multiple wavelength are considered, the variables relative to distinct wavelengths are not mixed together in the segregated groups.
- The **Thermal Damage** subfeature (added under **Biological Tissue** feature) defines an additional variable `alpha` that is placed in a dedicated segregated group.

DEFAULT NONLINEAR SETTINGS FOR MOISTURE TRANSPORT INTERFACES

Nonlinear solver settings depend on the study type.

Fully Coupled Solver Attribute

A fully coupled solver attribute is set by default. A Newton nonlinear method with a constant damping factor is set by default, with the following settings:

- The **Damping factor** is set to 0.7.
- The **Jacobian update** is set to **On first iteration** for **Time-Dependent** studies, and to **On every iteration** for **Stationary** studies or when the liquid concentration on surfaces, `cl_evap`, is solved.
- The **Maximum number of iterations** is set to 5 for **Time-Dependent** studies, and to 50 for **Stationary** studies.
- The **Tolerance factor** is set to 0.01 for **Time-Dependent** studies, and to 0.1 for **Stationary** studies.

Segregated Solver Attribute

The segregated solver attribute is set by default when another physics interface is solved together with moisture transport. The dependent variables of the Moisture Transport interface are placed in a separate segregated group.

A Newton nonlinear method with a constant damping factor is set by default, with the following settings:

- The **Damping factor** is set to 0.7.

- The **Jacobian update** is set to **On first iteration** for **Time-Dependent** studies, and to **On every iteration** for **Stationary** studies or when the liquid concentration on surfaces, cl_evap, is solved.

- The termination technique is **Iterations**, with **Number of iterations** set to 2.

DEFAULT NONLINEAR SETTINGS FOR THE LUMPED THERMAL SYSTEM INTERFACE

A fully coupled solver attribute is set by default. A Newton nonlinear method with a constant damping factor is set by default, with the following settings:

- The **Damping factor** is set to 1.

- The **Jacobian update** is set to **On every iteration**.

- The **Maximum number of iterations** is set to 5 for **Time-Dependent** studies, and to 50 for **Stationary** 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.

DEFAULT TIME 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 a **Thermal Damage** subfeature is present under **Biological Tissue** feature, particular settings for the time-dependent solver are used to efficiently compute the damage indicators:

- The **Absolute Tolerance** of the scaled damage indicator variable is set to 1, meaning that these variable are neglected in the error estimate.

- The damaged tissue indicator, α, is solved with an iterative **Jacobi** method.

- If the **Adaptive mesh refinement** option is selected in the study settings, the error indicator is set to $\sqrt{\nabla\theta_{d,\,sm} \cdot \nabla\theta_{d,\,sm}}$, where $\theta_{d,\,sm}$ 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 instant necrosis indicator, alphanecr, 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. It uses a direct linear solver. The default nonlinear method is the Newton method with constant damping factor.

When the **Irreversible Transformation** feature is active, similar settings are used:

- The **Absolute Tolerance** of the scaled irreversible transformation indicator dependent variable is set to 1.

- The irreversible transformation indicator α is solved with an iterative **Jacobi** method.

DEFAULT TIME SETTINGS FOR MOISTURE TRANSPORT INTERFACES

The default time-stepping method for the Moisture Transport interfaces is BDF at second order. It includes algebraic variable from the error estimate.

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 and moisture transport define 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 default Relative tolerance is set to 0.01 for **Time-Dependent** studies, and to 0.1 for **Stationary** studies.

- 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*

Guidelines for Solving Multiphysics Problems

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 Nonisothermal Flow, Electromagnetic Heating, Heat and Moisture Transport, and Moisture Flow interfaces.

Nonisothermal Flow
The Nonisothermal 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 gravity force accounted for 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, except that the presmoother and postsmoother in the GMRES iterative solver switches from SOR to SCGS.

When a strong coupling is assumed (gravity force accounted for in the flow interface), the default solver merges the temperature, pressure, and velocity groups. In this case, the solver corresponds to the default solver of the heat transfer interface. This is meant to suits well for nonisothermal flows in which natural convection dominates. See Default Linear Settings for Heat Transfer and Moisture Transport interfaces, Default Nonlinear Settings for Heat Transfer Interfaces, and Default Time Settings for Heat Transfer Interfaces for details.

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 Transfer with Surface-to-Surface Radiation

The **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling assumes a weak coupling between temperature and radiosity variables solved by a Heat Transfer interface and a Surface-to-Surface Radiation interface, respectively. The default solver contains dedicated segregated groups for temperature and radiosity variables.

Heat and Moisture Transport

The Heat and Moisture Transport multiphysics coupling controls the solver settings for the temperature and relative humidity variables.

The default solver merges the temperature and relative humidity groups. The solver corresponds to the default solver of the moisture transport interface. See Default Linear Settings for Heat Transfer and Moisture Transport interfaces, Default Nonlinear Settings for Moisture Transport Interfaces, and Default Time Settings for Moisture Transport Interfaces for details.

Moisture Flow

The Moisture Flow multiphysics coupling controls the solver settings for the flow (velocity and pressure) and the relative humidity variables.

When it assumes a weak coupling between the flow and the moisture interfaces (typically no gravity force accounted for in the flow interface), the default solver contains dedicated segregated groups for moisture and flow dependent variables. Each uses the default linear solver of the corresponding interface, except that the presmoother and postsmoother in the GMRES iterative solver switches from SOR to SCGS.

When a strong coupling is assumed (gravity force accounted for in the flow interface), the default solver merges the relative humidity, pressure, and velocity groups. In this case, the solver corresponds to the default solver of the moisture transport interface. See Default Linear Settings for Heat Transfer and Moisture Transport interfaces, Default Nonlinear Settings for Moisture Transport Interfaces, and Default Time Settings for Moisture Transport Interfaces for details.

Plotting and Evaluating Results in Layered Materials

When modeling heat transfer in thin layers, the thin layer's thickness is not explicitly represented in the geometry. However, a 1D extra dimension is attached to the boundary, via the selection of a layered material, and degrees of freedom are added through the thickness of the layered material. This concerns the **Thin Layer**, **Thin Film**, and **Fracture** nodes of the Heat Transfer interface, and all the features of the Heat Transfer in Shells interface.

Therefore, dedicated tools are available to plot and evaluate the results in the 1D extra dimension.

Plotting Along and Through the Layered Material

There are predefined plots to display the variables in the 1D extra dimension:

- The **Layered Material Slice** plot displays the variables along the layered material, for specified through-thickness locations in the 1D extra dimension.

- The **Through Thickness** plot displays the variables in the thickness direction, at specified points on the boundary

---

See Layered Material, Layered Material Slice, and Through Thickness in the *COMSOL Multiphysics Reference Manual* for details about the **Layered Material** dataset, and slice and through-thickness plots in layered materials.

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

Built-in Operators for Evaluation in the Layered Material

EVALUATION ALONG THE LAYERED MATERIAL

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

- `xdimTag` is the extra dimension tag. For example, it may be `slmat1_xdim`, when a single layer material (`slmat1`) is selected in the **Layered material** list of the **Thin Layer** node. This tag can be deduced from the **Selection** column of the **Equation View** subnode of the node applied on the thin layer, by clicking the **Show More Options** button ( ) on the **Model Builder** tool bar and selecting **Equation View**.

- `xd` is the coordinate in the extra dimension. It varies from 0 to `slmat1.th`, which is the total thickness of the layered material. By convention, `xd=0` corresponds to the downside of the boundary where the thin layer is defined, whereas `xd=slmat1.th` corresponds to its upside. 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 T to evaluate the temperature. There are others postprocessing variables defined on the extra dimension that can be found in the **Equation View** subnode of the node applied on the thin layer.
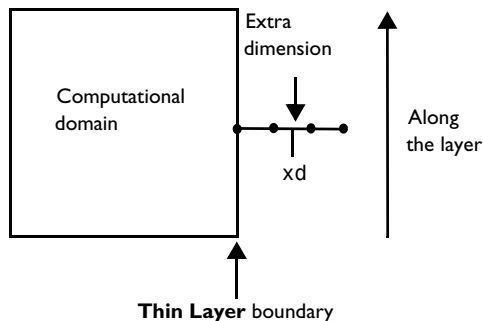
Figure 3-2: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results along the layer at the coordinate `xd`.

This operator can be used to define manually a slice plot in the layered material. In 2D and 2D axisymmetric geometries, the slice is represented in a line graph under a 1D plot group, whereas it is a surface plot for 3D geometries.

EVALUATION THROUGH THE LAYERED MATERIAL

The extra dimension temperature T can be evaluated through the layered material, at a specific point (x0, y0) or (x0, y0,z0), by using the operators atxd1 in 2D and 2D axisymmetric geometries, and atxd2 in 3D geometries. Note that this time the operators are used on the component and not on the extra dimension.

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 T 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 T to plot the temperature. Note that here too, these are 3D coordinates from the global coordinate system and not curvilinear coordinates.
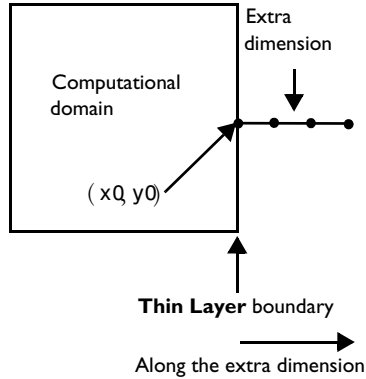
*Figure 3-3: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results through the layer at the point (*x0,y0*).*

This operator can be used to define manually a through thickness plot in the layered material. For all dimensions, the section is represented in a line graph under a 1D plot group. In order to use this, the **Dataset** selected in the **Data** section of the 1D plot group has to select the extra dimension as component. One method is to duplicate the default **Solution** node under the **Datasets** node, set the **Component** of the new node to the extra dimension, and use this new dataset into the 1D plot group, with the domains of the extra dimension selected.

INTEGRATION IN THE LAYERED MATERIAL

The integral of a variable in the layered material defined on a thin layer boundary can be evaluated by using the xdintopall operator, with the following expression: featTag.xdintopall(expr), where:

- featTag is the tag of the physics node defining the operator xdintopall. For example, for a **Thin Layer** node in the Heat Transfer interface, it may be ht.sls1.

- expr is the quantity to be integrated. For example, it can be set to T to evaluate the temperature.

LOCALIZATION OF THE EVALUATION IN THE THIN LAYER

For a layerwise computation, the atonly operator should be used to multiply the expression to be evaluated. This operator, applied to the extra dimension with the following expression, xdimTag.atonly(dom==i), returns 1 on the ith layer of the layered material, 0 on the other layers.

Here, `xdimTag` is the tag of the extra dimension. For example, it may be `llmat1_xdim` when a layered material link (`llmat1`) is selected in the **Layered material** list of the **Thin Layer** node. This tag can be deduced from the **Selection** column of the **Equation View** subnode of the node applied on the thin layer, by clicking the **Show More Options** button ( ) on the **Model Builder** tool bar and selecting **Equation View**.

Therefore, by using `featTag.xdintopall(xdimTag.atonly(dom==i)*expr)`, you evaluate the integral of `expr` on the ith layer of the layered material.

Q	See atxd, atonly, and noxd in the *COMSOL Multiphysics Reference Manual* for details about the operators.

Evaluating the Bulk Temperature

In a nonisothermal flow, the bulk temperature is an average temperature, taking into account the velocity field in the cross section of the flow. It can be evaluated in post processing, to estimate the heat transfer coefficient over a wall along the flow main direction.

This chapter describes how to evaluate the bulk temperature in 3D and 2D models, using the built-in geometry variables and operators. The nonisothermal flow is supposed to take place along a tube, with any orientation and curvature, even though the bulk temperature may be defined in other geometries. It contains the following sections:

- Bulk Temperature Definition
- Evaluation of the Bulk Temperature in 3D Models
- Evaluation of the Bulk Temperature in 2D Models

Bulk Temperature Definition

The bulk temperature T_b is defined as

$$T_b = \frac{\int_{S_\perp} (\mathbf{u} \cdot \tau) T ds}{\int_{S_\perp} \mathbf{u} \cdot \tau ds}$$

where $S_\perp$ is a cross-sectional surface of the tube where the flow takes place, $\mathbf{u}$ is the flow velocity field, τ is the tangential direction of the flow, and T is the temperature field.

The bulk temperature can be used to evaluate the heat transfer coefficient h at tube walls:

$$h = \frac{q_w}{T_w - T_b}$$

where q_w is the wall heat flux, and T_w is the wall temperature.

Evaluation of the Bulk Temperature in 3D Models

In 3D models, the bulk temperature may be evaluated along an edge of the tube in the direction of the flow, by using the built-in `diskint(R,expr,tol)` operator. When applied to an edge in 3D models, this operator integrates an expression `expr` over a disk of radius `R`, perpendicular to the selected edge. A numerical tolerance parameter `tol` is also available to adjust the numerical precision.

🔍   See ball, circle, disk, and sphere in the *COMSOL Multiphysics Reference Manual* for details about the operators.

In the evaluation of the bulk temperature, the radius `R` should be chosen large enough to encompass the tube cross section, but not too large regarding numerical precision.

If the evaluation disk overlaps other domains, the evaluation may be restricted to the domain of interest by adding a test on the `dom` variable.

The evaluation of the tangential velocity over the cross section requires to access the vector tangent to the edge, in the computational domain. However, the tangent vectors `t1` and `t2` are defined on boundaries only. By applying the general extrusion operator `genext1` on the edge, with a mapping to the domain, the tangential vector `genext1(t1)` is defined all over the cross section, and can be used in the `diskint` operator expression.

🔍   See General Extrusion in the *COMSOL Multiphysics Reference Manual* for details about the operators.

Finally, the bulk temperature can be expressed as

```
Tb =
diskint(R,if(dom==i,ut*T,0),tol)/diskint(R,if(dom==i,ut,0),tol)
```

where ut = u*genext1(t1x)+v*(genext1(t1y)+w*genext1(t1z), i is the domain entity number, and the tolerance `tol` should be specified.

Evaluation of the Bulk Temperature in 2D Models

In 2D models, the evaluation disk of the `diskint` operator is oriented in-plane, so this operator is not suitable for an integration over the cross section, and the `integrate(expr,var,lower,upper,tol)` operator should be used instead. This

operator computes the integral of `expr` for the integration variable `var` over an interval specified by expressions `lower` and `upper`. A relative tolerance `tol` may be specified as well.

	See integrate in the *COMSOL Multiphysics Reference Manual* for details about the operators.

In order to integrate over the cross section of the flow, the cross section line is parametrized as follows:

$$S_\perp = \{(x + sn_x, y + sn_y), s \in [0, 1]\}$$

where (n_x, n_y) is the normal vector to the boundary, and hence the vector tangent to the cross section.

The `at2` operator, which is the spatial evaluation operator in 2D, should be used to evaluate the expression `expr` on this parametrization of the cross section.

	See at in the *COMSOL Multiphysics Reference Manual* for details about the operators.

In practice, the integral is performed over a larger interval `[-R,R]`, where `R` should be chosen large enough to encompass the cross section of the tube, but not too large regarding numerical precision. The `isnan` operator can be used to ensure that the solution is defined at the prescribed coordinates, and as in 3D models, the `dom` variable may be used to identify the domain of interest.

	See isinf and isnan in the *COMSOL Multiphysics Reference Manual* for details about the operators.

As in 3D models, the evaluation of the tangential velocity over the cross section requires to access the vector tangent to the boundary, in the computational domain. However, the tangent vector `t` is defined on boundaries only. By applying the general extrusion operator `genext1` on the boundary, with a mapping to the domain, the

tangential vector `genext1(t)` is defined all over the cross section, and can be used in the `integrate` operator expression.

> See General Extrusion in the *COMSOL Multiphysics Reference Manual* for details about the operators.

Finally, the bulk temperature can be expressed as:

```
Tb = A/B
```

where

```
A = integrate(if(isnan(at2(x+s*nx,y+s*ny,ut*T)),0,at2(x+s*nx,y+s*
ny,ut*T)),s,-R,R,tol)
B = integrate(if(isnan(at2(x+s*nx,y+s*ny,ut*T)),0,at2(x+s*nx,y+s*
ny,ut)),s,-R,R,tol)
```

and `ut = u*genext1(tx)+v*(genext1(ty)`, `i` is the domain entity number, and the tolerance `tol` should be specified.

Ambient Properties

The ambient data available under the **Definitions** branch come from the processing of measured data from ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.) and ASHRAE Weather Data Viewer 6.0 (©2017 ASHRAE, www.ashrae.org. Used with permission.), given as frequencies of observations, monthly and hourly averaged for several past years of observation. They provide time-dependent weather conditions for more than 8000 stations worldwide in terms of the dry bulb temperature, the dew point temperature, the relative humidity, the wind speed, the absolute pressure, the precipitation rate, and the solar direct and diffuse irradiance.

In this section:

- Settings for the Ambient Properties
- Processing of ASHRAE Data
- Ambient Data Interpolation
- Ambient Variables and Conditions
- Use of Ambient Data From the Features

🔍	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, dew point temperature, and relative humidity.

Settings for the Ambient Properties

To add an **Ambient Properties** node to any Component, on the **Physics** toolbar click **Ambient Properties** or right-click the **Definitions** node under the Component and choose **Ambient Properties**.

In the **Ambient Properties** node (☀️), 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 direct and diffuse solar irradiances $I_{sn,amb}$ and $I_{sh,amb}$.

> ℚ
> See Heat Flux, Diffuse Surface (Surface-to-Surface Radiation Interface), 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.

Three options are available for the definition of the **Ambient data**:

- When **User defined** (the default) is selected, the **Temperature** T_{amb}, the **Absolute pressure** p_{amb}, the **Relative humidity** ϕ_{amb}, the **Wind velocity** v_{amb}, the **Precipitation rate** $P_{0,amb}$, the **Clear sky noon beam normal irradiance** $I_{sn,amb}$, and the **Clear sky noon diffuse horizontal irradiance** $I_{sh,amb}$ should be specified directly.

- When either **Meteorological data (ASHRAE 2013)** or **Meteorological data (ASHRAE 2017)** is selected, the ambient variables are computed from monthly and hourly averaged measurements, made over several years at weather stations worldwide. See Processing of ASHRAE 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 8000 weather stations worldwide. Two options are available for the selection of the **Weather station**:

- When **From list** is selected, click on the **Set Weather Station** button to open the **Weather Station** browser that allows you to select a **Region**, a **Country**, and a **Station**.

The station details (World Meteorological Organization reference, location, and geographical coordinates) are displayed underneath.

- It is also possible to select a station **From reference** by entering the corresponding **World Meteorological Organization reference**, which is a 6-digit number.

Q	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.

Q	When a station is selected **From list**, its World Meteorological Organization (WMO) reference is displayed in the user interface. 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/v2.0/.

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 **On** is selected in the **Specify year** list, 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 start 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. Clear this check box to manually set the time update.

Q	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.

Ambient Conditions

Based on the measured data, several conditions are available for the **Temperature**, the **Dew point temperature**, the **Wind speed**, and the **Precipitation rate**. The formula for each condition is recalled in Table 3-11, Table 3-12, and Table 3-13. 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 3-11: 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 3-13.

TABLE 3-12: 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 3-13: WIND SPEED CONDITIONS

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

TABLE 3-14: PRESSURE CONDITIONS

CONDITION	DEFINITION
Average	$P_{0,\text{amb}} = \langle P_{0,\text{station}} \rangle$
Low	$P_{0,\text{amb}} = \langle P_{0,\text{station}} \rangle - \sigma_{P_{0,\text{station}}}$
High	$P_{0,\text{amb}} = \langle P_{0,\text{station}} \rangle + \sigma_{P_{0,\text{station}}}$
Lowest	$P_{0,\text{amb}} = \min(P_{0,\text{station}})$
Highest	$P_{0,\text{amb}} = \max(P_{0,\text{station}})$

	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.

	The sum of the **Clear sky noon beam normal irradiance** and the **Clear sky noon diffuse horizontal irradiance** is available through the postprocessing variable ht.Is_amb, defined as the **Ambient solar irradiance**.

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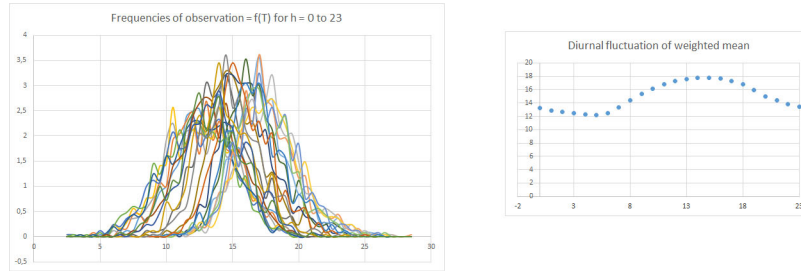
Figure 3-4: Computation of weighted mean from frequencies of observations for the diurnal fluctuations of temperature.

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 interpolation provides:

- The annual fluctuation of the dew point temperature, the relative humidity, the wind speed, and the direct and diffuse solar irradiances.
- 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:

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

- Low:

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

- High:

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

- Lowest:

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

- Highest:

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

- User defined coefficient for deviation:

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

- User defined correction:

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

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 1$^{\text{st}}$ of June.
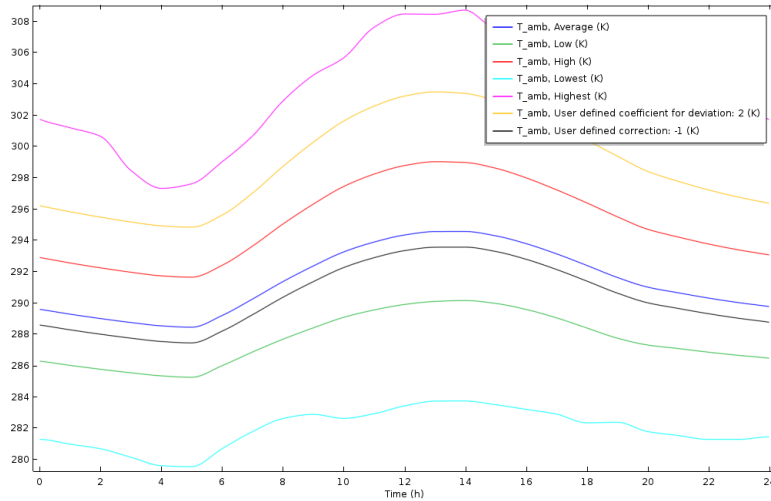
Figure 3-5: Comparison of ambient conditions for the temperature at New York/John F. Ke, on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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

- Heating wind correlation:

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

- Cooling wind correlation:

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

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

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

where $\Delta T_{\text{ws,station}}$ (SI unit: K) and $\Delta T_{\text{wd,station}}$ (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 1$^{\text{st}}$ of June.
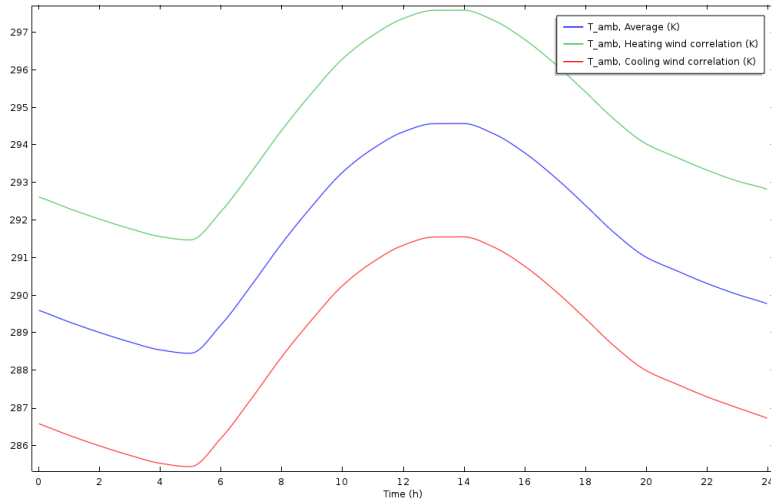
Figure 3-6: Comparison of heating and cooling wind correlations for the temperature at New York/John F. Ke, on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. *Used with permission.)*

CONDITIONS OF DEW POINT TEMPERATURE

- Average:

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

- Low:

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

- High:

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

- Lowest:

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

- Highest:

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

where:

- $<DPT_{\text{station}}>$ (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.
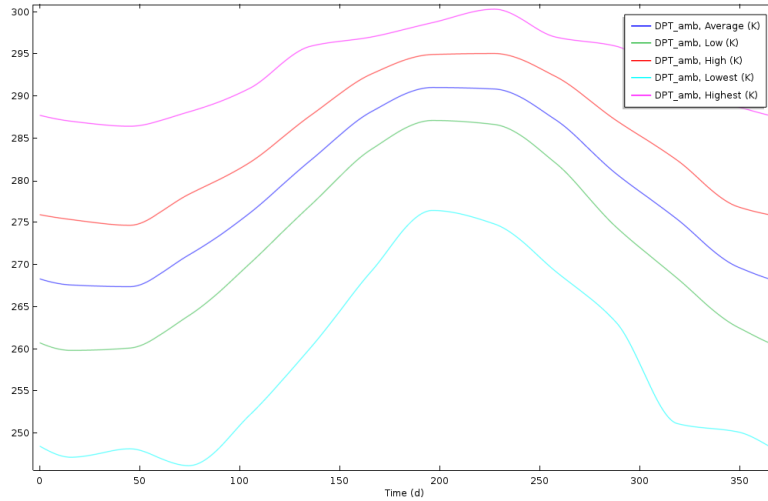
Figure 3-7: Comparison of the ambient conditions for the dew point temperature at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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:

- $\langle v_{\text{station}} \rangle$ (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.
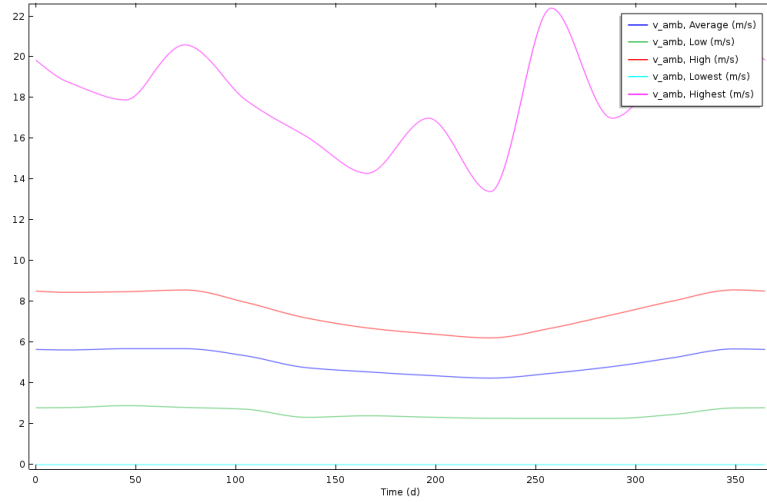
Figure 3-8: Comparison of the ambient conditions for the wind speed at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

CONDITIONS OF PRECIPITATION RATE

• Average:

$$P_{0,\,amb} = \langle P_{0,\,station} \rangle$$

• Low:

$$P_{0,\,amb} = \langle P_{0,\,station} \rangle - \sigma_{P_{0,\,station}}$$

• High:

$$P_{0,\,amb} = \langle P_{0,\,station} \rangle + \sigma_{P_{0,\,station}}$$

• Lowest:

$$P_{0,\,amb} = \min(P_{0,\,station})$$

• Highest:

$$P_{0,\,amb} = \max(P_{0,\,station})$$

where:

- $<P_{0,\text{station}}>$ (SI unit: m/s) is the weighted mean of the observed values of precipitation rate at the station.

- $\sigma_{P0,\text{station}}$ (SI unit: m/s) is the standard deviation of the observed values of precipitation rate at the station.

- $P_{0,\text{station}}$ (SI unit: m/s) is the set of the observed values of precipitation rate at the station.

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:

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

where $p_{\text{sat}}(T_{\text{amb}})$ 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.
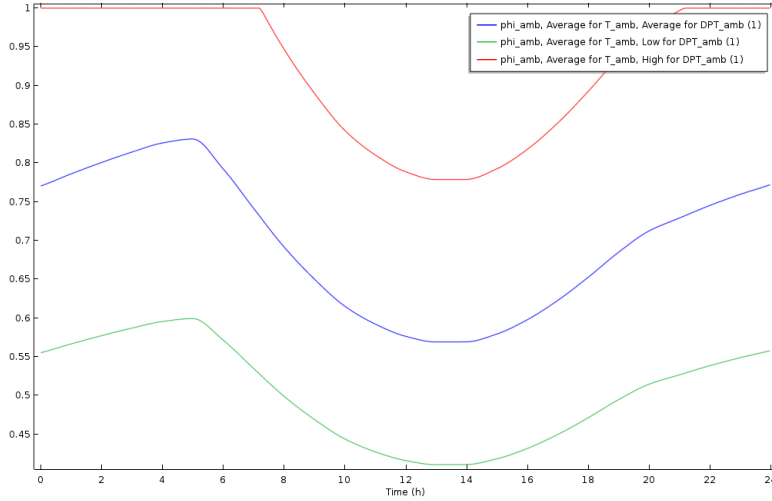
Figure 3-9: Diurnal fluctuations of relative humidity for different ambient conditions at New York/John F. Ke, on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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:

$$x_{vap, amb} = \frac{\phi_{amb} \cdot p_{sat}(T_{amb})}{p_{amb} - \phi_{amb} \cdot p_{sat}(T_{amb})} \cdot \frac{M_v}{M_a}$$

where $p_{sat}(T_{amb})$ 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.

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. The sum of direct and diffuse solar irradiances is the ambient solar irradiance $I_{s,\,amb}$ (SI unit: W/m^3), defined by:

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

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

Figure 3-10 illustrates the evolution of ambient solar irradiance for New York/John F. Ke, over the year.
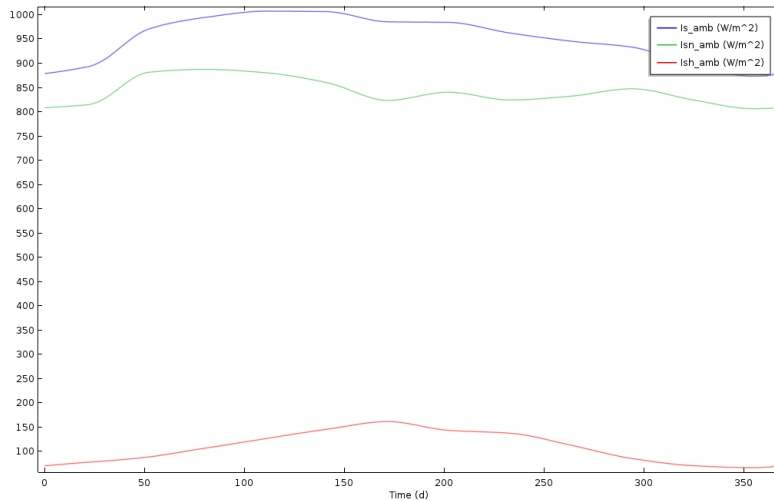
Figure 3-10: Decomposition of solar irradiance into normal and horizontal irradiance at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

Use of Ambient Data From the Features

The ambient variables defined in the **Ambient Properties** node 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**

Modeling Heat Transfer by Radiation

This section summarizes the functionalities available within the Heat Transfer Module for the modeling of heat transfer by radiation.

Radiation is a heat transfer mode that should be considered in addition to conduction and convection in the following cases:

- In a transparent or semitransparent medium.

- For a surface with a large emissivity (close to 1). Whereas most polished metals have an emissivity below 0.1, other materials such as concrete, black painting, glass, plastic, skin, paper, and ceramic have an emissivity over 0.8 for infrared radiation.

- When the temperature difference between a surface and the surrounding surfaces or the ambient is large.

- When conduction and convection are small, as radiation becomes the predominant transfer mode.

- At small time scales, for which conduction and convection effects are too slow and have a less significant effect.

As an example, consider a plate of concrete with a cold bottom wall at temperature T_0, placed in a hot environment with an air flux at temperature T_{amb}:
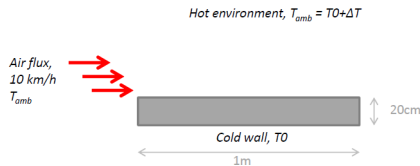
The figures below illustrate the relative importance of heat transfer by radiation and by convection (due to the air flux), when the emissivity ε is 0.1 (left) and 0.9 (right).
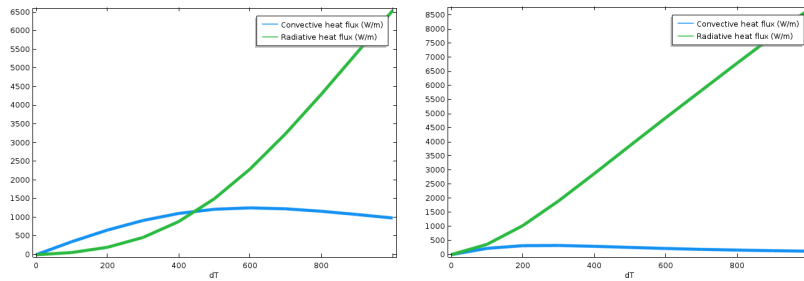
Figure 3-12: Integral of convective and radiative heat fluxes (W/m) along the top boundary, for two values of emissivity, as a function of the temperature difference.

Whereas the boundary radiative and convective heat fluxes are of the same order for temperature difference up to 500K when the emissivity is low (ε=0.1), radiation becomes the dominant mode of heat transfer even for a small temperature difference when the emissivity is high (ε=0.9). Note that the convective heat flux decrease observed for high temperature gradients is related to the fact that the velocity, not the mass flow rate, is prescribed at the air inlet.

> See Heat Transfer Variables for the definition of the variables q0 and rflux giving access to the convective and radiative heat fluxes on boundaries.

Two configurations are considered regarding the functionalities available for the modeling of radiative heat transfer:

• Radiation in transparent media

• Radiation in participating media

Interfaces and Features for Radiation in Transparent Media

The following table recalls the functionalities available for the modeling of radiative heat transfer in transparent media.

TABLE 3-15: MODELING OF HEAT TRANSFER BY RADIATION IN TRANSPARENT MEDIA

	SURFACE-TO-AMBIENT	SURFACE-TO-SURFACE
Description		
Physical and modeling assumptions		
Refractive index of domain	n=1	Any, uniform
Diffuse reflection	Yes	Yes
Specular reflection	No	With Ray shooting
Transmission	No	With Ray shooting
Shadowing effects in view factor computation	No	With Hemicube or Ray shooting
Wavelength dependence of surface properties	Yes (Solar and ambient / Multiple spectral bands) No (Constant)	Yes (Solar and ambient / Multiple spectral bands) No (Constant)
Directional dependence of surface properties	No	With Ray shooting
Dimension of the domain	3D, 2D axi, 2D, 1D axi, 1D	3D, 2D axi, 2D, 1D axi, 1D
Example in Application Library	Continuous Casting	Chip Cooling
License, interfaces and features		
License requirement	COMSOL Multiphysics / Heat Transfer Module	Heat Transfer Module
Interfaces	Heat Transfer (ht) Heat Transfer in Shells (htlsh)	Surface-to-Surface Radiation (rad)
Domain features	Out-of-Plane Radiation (1)(2)	Opacity

TABLE 3-15: MODELING OF HEAT TRANSFER BY RADIATION IN TRANSPARENT MEDIA

	SURFACE-TO-AMBIENT	SURFACE-TO-SURFACE
Boundary features	Surface-to-Ambient Radiation	Diffuse Mirror
		Diffuse Surface
		Opaque Surface
		Semitransparent Surface
		Initial Values
		Prescribed Radiosity
		Radiation Group
Edges/Point features	Surface-to-Ambient Radiation[1]	N/A
Global features	N/A	External Radiation Source [3]
		Symmetry for Surface-to-Surface Radiation [4]
Multiphysics features	N/A	Heat Transfer with Surface-to-Surface Radiation
Numerical aspects		
Numerical cost	Low	Medium to high
Radiation variables	None	Surface radiosity J (if constant wavelength), J_1,J_2 (if solar and ambient wavelengths),
		$J_1,...,J_n$ (if multiple wavelengths)

[1] Only with the Heat Transfer Module

[2] Only in 2D and 1D components

[3] Only in 3D and 2D components

[4] Only in 3D, 2D axi, and 2D components

[5] Only with Rosseland approximation and P1 method

The following table recalls the functionalities available for the modeling of radiative heat transfer in participating media.

TABLE 3-16: MODELING OF HEAT TRANSFER BY RADIATION

	ABSORBING MEDIA	ABSORBING-SCATTERING MEDIA	PARTICIPATING MEDIA
Description			
Physical and modeling assumptions			
Refractive index of domain	Any, uniform	Any, uniform	Any, uniform
Scattering in domain	N/A	Isotropic (P1 method, DOM) Linear anisotropic (P1 method, DOM) Anisotropic (DOM) Henyey-Greenstein (DOM)	Isotropic (Rosseland approximation,P1 method, DOM) Linear anisotropic (P1 method, DOM) Anisotropic (DOM) Henyey-Greenstein (DOM)
Wavelength dependency of emissivity	No	Yes (Solar and ambient / Multiple spectral bands)	Yes (Solar and ambient / Multiple spectral bands) No (Constant)
Dimension of the domain	3D, 2D axi[5], 2D	3D, 2D axi[5], 2D	3D, 2D axi[5], 2D
Example model	N/A	N/A	Glass Plate
License, interfaces and features			
License requirement	Heat Transfer Module	Heat Transfer Module	Heat Transfer Module

TABLE 3-16: MODELING OF HEAT TRANSFER BY RADIATION

	ABSORBING MEDIA	ABSORBING-SCATTERING MEDIA	PARTICIPATING MEDIA
Interfaces	Radiative Beam in Absorbing Media (rbam)	Radiation in Absorbing-Scattering Media (rasm)	Radiation in Participating Media (rpm) Heat Transfer (ht): Rosseland approximation
Domain features	Absorbing Medium, Initial Values	Absorbing-Scattering Medium, Initial Values, Radiative Source	Participating Medium, Initial Values, Radiative Source Optically Thick Participating Medium
Boundary features	Opaque Surface Incident Intensity Transparent Surface	Opaque Surface Incident Intensity Continuity of Interior Boundary (DOM) Periodic Condition Symmetry (P1 method)	Opaque Surface Semitransparent Surface Incident Intensity Continuity of Interior Boundary (DOM) Periodic Condition Symmetry (P1 method)
Multiphysics features	Heat Transfer with Radiative Beam in Absorbing Media (htrbam)	Heat Transfer with Radiation in Absorbing-Scattering Media (htrasm)	Heat Transfer with Radiation in Participating Media (htrpm)
Numerical aspects			
Numerical cost	Medium	Medium (P1 method) High (DOM)	Low (Rosseland approximation) Medium (P1 method) High (DOM)

TABLE 3-16: MODELING OF HEAT TRANSFER BY RADIATION

	ABSORBING MEDIA	ABSORBING-SCATTERING MEDIA	PARTICIPATING MEDIA
Radiation variables	Incident intensities I_i	Incident radiation G (P1 method) Radiative intensity $I=I_1,...,I_n$, with n up to 512 (DOM)	None (Rosseland approximation) Incident radiation G (P1 method) Radiative intensity $I=I_1,...,I_n$, with n up to 512 (DOM)

[1] Only with the Heat Transfer Module

[2] Only in 2D and 1D components

[3] Only in 3D and 2D components

[4] Only in 3D, 2D axi, and 2D components

[5] Only with Rosseland approximation and P1 method

Specifying the Temperature at a Flow Inlet

When modeling a nonisothermal flow, a **Temperature** condition can be used at a flow inlet where the temperature of the fluid is a known quantity. However, there are some other situations where this is not the case, and an **Inflow** boundary condition can improve the model accuracy and reduce the computational cost of the simulation.

In this section:

- Using the Temperature Condition
- Using the Inflow Condition
- Temperature Contribution to the Inflow Boundary Condition
- Pressure Contribution to the Inflow Boundary Condition
- Choosing Between the Temperature and the Inflow Conditions

Using the Temperature Condition

The **Temperature** condition is available at exterior and interior boundaries of any type of domain and is equivalent to set the temperature to a known value on the selected boundary.

Using the Inflow Condition

The **Inflow** condition is available at boundaries that are exterior to a fluid domain and is equivalent to having a virtual channel "upstream". It is used to define a heat flux at the inlet boundary that brings the same energy to the fluid domain as if you had modeled the virtual channel as a real CFD domain. The virtual channel can be seen as

a long insulated channel with given thermal properties at the inlet, and with the same velocity profile as defined in the settings for the **Inflow** boundary condition.
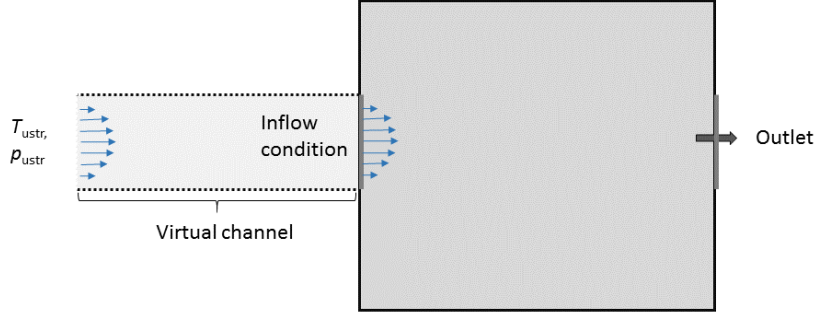
Figure 3-13: Representation of the virtual domain corresponding to an Inflow boundary condition.

This is of particular interest when the geometry contains inlets that are fed by channels that are not represented in the geometry.

From a mathematical point of view, the boundary condition is formulated as a flux condition:

$$-\mathbf{n} \cdot \mathbf{q} = \rho \Delta H \mathbf{u} \cdot \mathbf{n}$$

where the enthalpy variation ΔH depends on temperature and pressure difference:

$$\Delta H = \int_{T_{ustr}}^{T} C_p dT + \int_{p_{ustr}}^{p} \frac{1}{\rho}(1 - \alpha_p T)dp = \Delta H_T + \Delta H_p \tag{3-1}$$

where:

$$\Delta H_T = \int_{T_{ustr}}^{T} C_p dT \tag{3-2}$$

and

$$\Delta H_p = \int_{p_{ustr}}^{p} \frac{1}{\rho}(1 - \alpha_p T)dp \tag{3-3}$$

Equation 3-1 expresses the fact that the normal conductive heat flux at the inflow boundary is proportional to the flow rate and enthalpy variation between the upstream conditions and inlet conditions.

Temperature Contribution to the Inflow Boundary Condition

As shown in Equation 3-1, the enthalpy variation depends in general both on the difference in temperature and in pressure. However, the pressure contribution to the enthalpy, ΔH_p, can be neglected when the work due to pressure changes is not included in the energy equation. This is controlled by the **Nonisothermal Flow** multiphysics coupling depending on compressibility assumption.

There is another classical case where this term cancels out: when the fluid is modeled as an ideal gas. Indeed, in this case,

$$\alpha_p = \frac{1}{T}$$

When the pressure contribution to the enthalpy is neglected, the boundary condition reads:

$$k\nabla T \cdot \mathbf{n} = \rho\left(\int_{T_{\text{ustr}}}^{T} C_p dT\right)\mathbf{u} \cdot \mathbf{n} \tag{3-4}$$

When advective heat transfer dominates at the inlet (large flow rates), the temperature gradient, and hence the heat transfer by conduction, in the normal direction to the inlet boundary is very small. So in this case, Equation 3-4 imposes that the enthalpy variation is close to zero. As C_p is positive, the **Inflow** boundary condition requires $T=T_{\text{ustr}}$ to be fulfilled. So, when advective heat transfer dominates at the inlet, the **Inflow** boundary condition is almost equivalent to a Dirichlet boundary condition that prescribes the upstream temperature at the inlet.

Conversely, when the flow rate is low or in the presence of large heat sources or sinks next to the inlet, the conductive heat flux cannot be neglected. In addition, the inlet temperature has to be adjusted to balance the energy brought by the flow at the inlet and the energy transferred by conduction from the interior, as described by Equation 3-4. This makes it possible to observe a realistic upstream feedback due to thermal conduction from the inlet surroundings.

Pressure Contribution to the Inflow Boundary Condition

In the general case, the enthalpy variation depends both on the difference in temperature and in pressure.

Considering that the **Inflow** boundary condition models a virtual channel feeding the inlet, pressure losses are expected between the virtual channel inlet and the boundary where the condition is defined. This explains why the upstream pressure is different from the inlet pressure. While the fluid flows through the channel, it is subject to pressure work that results in a temperature change between the virtual channel inlet and the boundary where the **Inflow** boundary condition is defined. This is what is described by the pressure-dependent term in Equation 3-1. Note that the viscous dissipation in the virtual channel is not accounted for.

Choosing Between the Temperature and the Inflow Conditions

While in many configurations, the **Temperature** and **Inflow** features describe similar conditions and lead to similar simulation results, there are a number of configurations (especially for slow flow and small dimensions) where the conductive effects are not dominated by the advective effects and where the **Inflow** boundary condition usually leads to a temperature profile that is closer to the reality than a **Temperature** boundary condition. In these cases, a **Temperature** boundary condition may enforce an erroneous temperature value that induces large heat fluxes that are not realistic.

Heat Part Library

The Part Libraries can be used to store and access a set of standard geometric entities. It is possible to create custom user-defined geometry libraries or to use built-in collections of geometric entities that are available with many add-on modules for COMSOL Multiphysics.

Part Libraries in the *COMSOL Multiphysics Reference Manual*

In this section:

- Heat Part Library Contents
- Heat Sink — Parameterized Fin Types Part
- Heat Sink — Dissimilar Border Pins Part
- Heat Sink — Pin Fins Part
- Heat Sink — Straight Fins Part

Heat Part Library Contents

The part library included with the Heat Transfer Module contains typical heat sink geometries. When accessing the library, the parts are sorted as in the figure below:
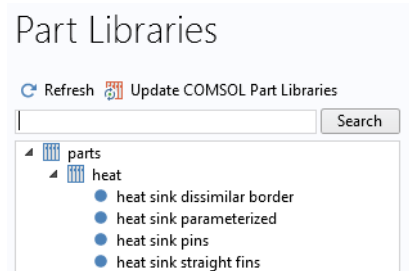
The heat sinks are composed of a rectangular base and an array of pin or straight fins, as shown on Figure 3-14.
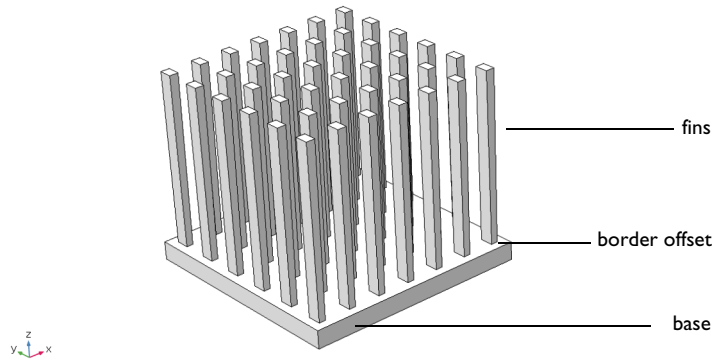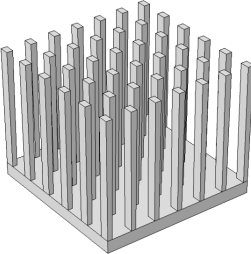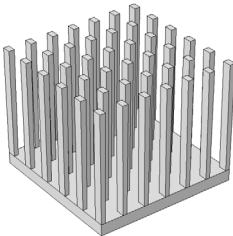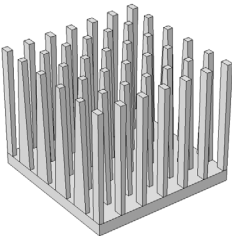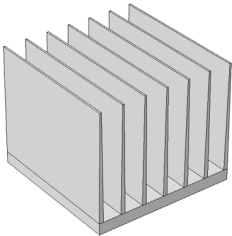
Figure 3-14: Heat sinks components

All entities are fully parameterized, making them easy to use as parts in industrial models where heat sinks are used for cooling. For example, you can control the number, the shape, the dimensions, and the placement of the fins on the base. In addition, fillet, chamfer, and notch transformations can be applied to the fins, and parameter checks are applied to ensure that the values set in the **Input Parameters** section are valid. Finally, the fins can be defined as solids or as boundaries for computational efficiency.

The library includes four parts. The **Heat Sink — Parameterized Fin Types** part is the most general part. It supports multiple parameterization, while a reduced number of parameters is available in the three other parts, as shown in Table 3-17.

TABLE 3-17: OVERVIEW OF HEAT SINK PARTS

Heat Sink — Parameterized Fin Types

fin_type =1,2,3 (editable)

Heat Sink — Pin Fins	Heat Sink — Dissimilar Border Pins	Heat Sink — Straight Fins
fin_type = 1 (not editable)	fin_type = 2 (not editable)	fin_type = 3 (not editable)

In the **Heat Sink — Pin Fins** part, all the fins are pins with the same dimension, whereas the outer and inner fins (in the y direction) can have distinct dimensions in the **Heat Sink — Dissimilar Border Pins** part. You may use the **Heat Sink — Straight Fins** part to define a heat sink made of only straight fins.

By default, the base of the heat sink is positioned at the origin of the xy-plane. You can apply a **Displacement** and a **Rotation** to this configuration in the **Position and Orientation of Output** section.

Finally, each fin is a separate domain and can be easily deleted, and selections can be defined, for example for the base bottom face (**Boundary Selections** section) or the array of fins (**Domain Selections** section).

Heat Sink — Parameterized Fin Types Part

The dimensions of the base and the fins can be parameterized, as shown on Figure 3-15. In addition, an offset from the base border can be added.
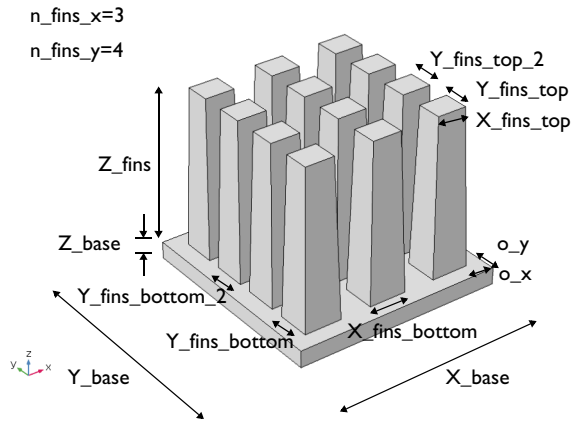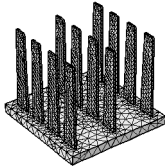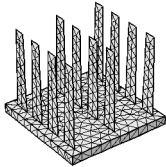
Figure 3-15: Fins and base parameters in Heat Sink — Parameterized part

3D AND SHELL FINS

When the thickness of the fins is small in the x direction, the fins can be defined as boundaries instead of solids in order to reduce the size of the model. Indeed, small elements are required to mesh the fins across their thickness in this case. It is done by setting the shell parameter to 1. The default meshes obtained with shell = 0 (left)

and `shell=1` (right) are shown in Table 3-18 for an example configuration. Setting `shell` to 1 significantly reduces the number of mesh elements, as shown below.

TABLE 3-18: DEFAULT TETRAHEDRAL MESH WITH 3D FINS AND SHELL FINS

SHELL=0	SHELL=1
Mesh statistics:	Mesh statistics:
8003 tetrahedra	906 tetrahedra
5900 triangles	1058 triangles
1220 edge elements	312 edge elements
104 vertex elements	56 vertex elements

Note that when `shell=1`, the `X_fins_bottom`, `Y_fins_bottom`, `X_fins_top`, and `Y_fins_top` parameters are not used.

STEP

A step can be defined in the x direction at the center on the base. The parameter `step_width` specifies the number of filled gaps from middle to border by the step, as shown on Figure 3-16. This option is not available for shell fins (`shell=1`).
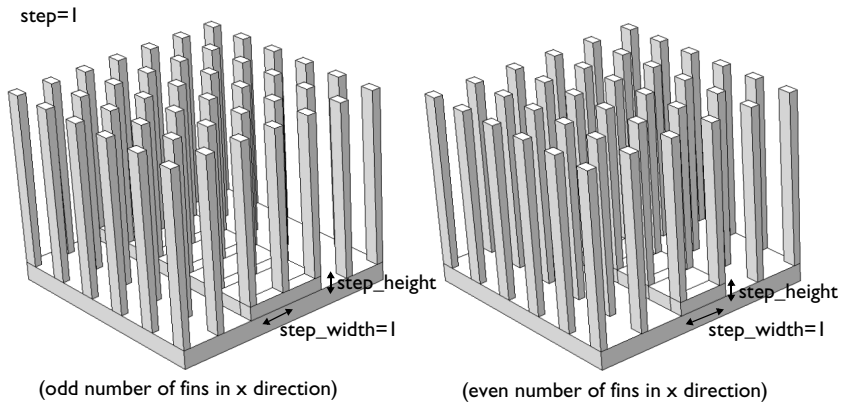
Figure 3-16: Step parameters in Heat Sink — Parameterized part

FILLET

Finally, a fillet transformation can be applied at the top and bottom of the fins, as shown on Figure 3-17. The fillet transformation at the bottom of the fins is not available for shell fins (`shell=1`).
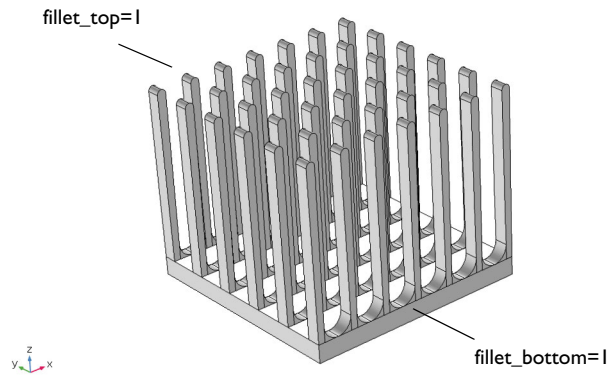
Figure 3-17: Fillet parameters in Heat Sink — Parameterized part

Heat Sink — Dissimilar Border Pins Part

This part inherits all the functionalities of the **Heat Sink — Parameterized Fin Types** part, but different dimensions are set by default for the inner and outer fins in the *y* direction:

- `Y_fins_bottom` = 3 mm (outer fins) and `Y_fins_bottom_2` = 6 mm (inner fins)
- `Y_fins_top` = 3 mm (outer fins) and `Y_fins_top_2` = 6 mm (inner fins)

Heat Sink — Pin Fins Part

This part inherits all the functionalities of the **Heat Sink — Parameterized Fin Types** part, but identical dimensions are enforced for the inner and outer fins in the *y* direction:

- `Y_fins_bottom` = 3 mm (all fins)
- `Y_fins_top` = 3 mm (all fins)

Heat Sink — Straight Fins Part

This part inherits all the functionalities of the **Heat Sink — Parameterized Fin Types** part, but the *y* direction parameters are enforced to define straight fins.

In addition, notch and chamfer transformations can be applied to the fins, as shown on Figure 3-18.
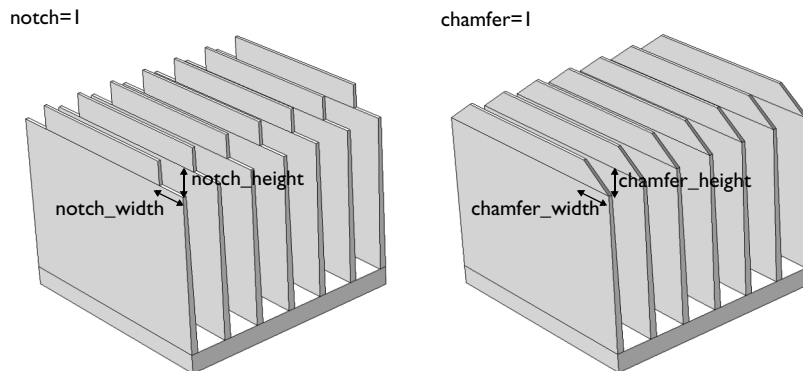
Figure 3-18: Notch and chamfer parameters in Heat Sink — Straight Fins part

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:

- Foundations of the General Heat Transfer Equation
- Theory for Heat Transfer in Solids
- Theory for Heat Transfer in Fluids
- Theory for Heat Transfer in Moist Air
- Theory for Bioheat Transfer
- Theory for Heat Transfer in Porous Media
- Theory for Heat Transfer with Phase Change
- Theory for Heat Transfer in Building Materials
- Theory for Harmonic Heat Transfer
- Theory for Lumped Isothermal Domain
- Theory for Heat Transfer in Thin Structures
- Theory for the Lumped Thermal System Interface
- Theory for Surface-to-Surface Radiation

- Theory for Radiation in Participating Media
- Theory for Moisture Transport
- Theory for the Heat Transfer Multiphysics Couplings
- Theory for Thermal Contact
- Out-of-Plane Heat Transfer
- Convective Heat Transfer Correlations
- Nucleate Pool Boiling Correlation
- Equivalent Thermal Conductivity Correlations
- Temperature Dependence of Surface Tension
- 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
- The Physical Mechanisms Underlying Heat Transfer
- The Heat Balance Equation
- Consistency with Mass and Momentum Conservation Laws

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:

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

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

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

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

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):

$$H = E - \frac{1}{\det(\mathbf{F})\rho} \mathbf{P}{:}\mathbf{F}$$

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:

$$dE = \left(\frac{\partial E}{\partial S}\right)_v dS + \left(\frac{\partial E}{\partial v}\right)_S dv$$

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

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

These lead to the fundamental thermodynamic relation:

$$dE = TdS - pdv$$

> 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:

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

$$dE = TdS + \frac{1}{\det(\mathbf{F})\rho}\mathbf{P}:d\mathbf{F}$$

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

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

and for a solid, the definitions become:

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

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:

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

$$\left(\frac{\partial H}{\partial T}\right)_{\mathbf{P}} = C_p \qquad \left(\frac{\partial H}{\partial \mathbf{P}}\right)_T = \frac{1}{\det(\mathbf{F})\rho}\mathbf{F}(-\mathbf{I} + \alpha T)$$

SENSIBLE ENTHALPY

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

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

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} (1 atm) and T_{ref} (293.15 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.

Sensible Enthalpy in Ideal Gas
In an ideal gas,

$$\alpha_p \;=\; \frac{1}{T}$$

therefore

$$\left(\frac{\partial H}{\partial p}\right)_T \;=\; 0$$

and Equation 4-5 reduces to

$$H \;=\; H_{ref} + \int_{T_{ref}}^{T} C_p \cdot dT \tag{4-6}$$

Sensible Enthalpy in Moist Air
The ideal gas assumption also holds for moist air, so

$$\left(\frac{\partial H}{\partial p}\right)_T \;=\; 0$$

In addition, by using a mixture formula, we have

$$\left(\frac{\partial H}{\partial T}\right)_p \;=\; (C_{p,\,v} - C_{p,\,a})\frac{\partial \omega_v}{\partial c_v}$$

and Equation 4-6 should be modified to include an integral over the vapor concentration:

$$H \;=\; H_{ref} + \int_{T_{ref}}^{T} (C_{p,\,v} - C_{p,\,a})\frac{\partial \omega_v}{\partial c_v} \cdot dT + \int_{c_{ref}}^{c_v} \frac{\partial H}{\partial c_v} \cdot dc_v \tag{4-7}$$

The Physical Mechanisms Underlying 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:

$$\mathbf{q} = -k\nabla T \tag{4-8}$$

  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):

$$\frac{dE_\Omega}{dt} + \frac{dK_\Omega}{dt} = P_{\text{ext}} + Q_{\text{exch}} \tag{4-9}$$

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):

$$\frac{dK_\Omega}{dt} + P_{\text{str}} = P_{\text{ext}} \tag{4-10}$$

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 4-10.

Combining Equation 4-9 and Equation 4-10 yields the so-called heat balance equation (2.3.65 in Ref. 4):

$$\frac{dE_\Omega}{dt} = P_{\text{str}} + Q_{\text{exch}} \tag{4-11}$$

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 4-10 and Equation 4-11 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 4-11.

LOCALIZED FORM

In this paragraph, the different terms of Equation 4-11 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):

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

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:

$$\sigma = -p\mathbf{I} + \tau \tag{4-12}$$

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

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

Equivalently, the stress power can also be expressed as:

$$P_{str} = \int_{\Omega} \frac{1}{\det(\mathbf{F})} \left(\mathbf{P} : \frac{d\mathbf{F}}{dt} \right) dv$$

Exchanged Heat

Finally, the exchanged heat rates, Q_{exch}, account for thermal conduction (see Fourier's Law at Equation 4-8), 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:

$$Q_{exch} = -\int_{\partial\Omega} (\mathbf{q} \cdot \mathbf{n})ds - \int_{\partial\Omega} (\mathbf{q}_r \cdot \mathbf{n})ds + \int_{\Omega} Qdv$$

Recall the following notations used above: $\mathbf{q}$ for the heat flux by conduction, $\mathbf{q}_r$ for the heat flux by radiation, Q for additional heat sources, and $\mathbf{n}$ for the external normal vector to the boundary $\partial\Omega$.

Localized Heat Balance Equation

With all these elements, the heat balance equation (Equation 4-11) becomes:

$$\int_{\Omega} \rho \frac{dE}{dt} dv + \int_{\partial\Omega} (\mathbf{q} \cdot \mathbf{n})ds + \int_{\partial\Omega} (\mathbf{q}_r \cdot \mathbf{n})ds = \int_{\Omega} (\sigma : \mathbf{D})dv + \int_{\Omega} Qdv \tag{4-13}$$

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

$$\rho \frac{dE}{dt} + \nabla \cdot (\mathbf{q} + \mathbf{q}_r) = \sigma : \mathbf{D} + Q \tag{4-14}$$

or equivalently in the *spatial* frame:

$$\rho \frac{\partial E}{\partial t} + \rho \mathbf{u} \cdot \nabla E + \nabla \cdot (\mathbf{q} + \mathbf{q}_r) = \sigma : \mathbf{D} + Q \tag{4-15}$$

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 4-15 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 4-15 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	$\sigma^{\mathrm{T}} = \sigma$	$\sigma^{\mathrm{T}} = \sigma$

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 4-15:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u}_{\text{trans}} \cdot \nabla T \right) + \nabla \cdot (\mathbf{q} + \mathbf{q}_r) = -\alpha T \colon \frac{dS}{dt} + Q \tag{4-16}$$

The different quantities involved here are recalled below:

- ρ is the density (SI unit: 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: W/m^2)
- $\mathbf{q}_r$ is the heat flux by radiation (SI unit: 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: 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 4-16 is the *thermoelastic damping* and accounts for thermoelastic effects in solids:

$$Q_{\text{ted}} = -\alpha T \colon \frac{dS}{dt} \tag{4-17}$$

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):

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot (\mathbf{q} + \mathbf{q}_r) = \alpha_p T \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right) + \tau : \nabla \mathbf{u} + Q \qquad (4\text{-}18)$$

which is derived from Equation 4-15, 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: 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: W/m^2)
- $\mathbf{q}_r$ is the heat flux by radiation (SI unit: W/m^2)
- α_p is the coefficient of thermal expansion (SI unit: 1/K):

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

  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: 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 4-18 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.

$$Q_p = \alpha_p T\left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p\right) \tag{4-19}$$

The second term represents viscous dissipation in the fluid:

$$Q_{\mathrm{vd}} = \tau : \nabla \mathbf{u} \tag{4-20}$$

Theory for the Inflow Boundary Condition

At the inlet boundary of a fluid domain, the Inflow boundary condition defines a heat flux that accounts for the energy that would normally be brought by the fluid flow if the channel upstream to the inlet was modeled.

DANCKWERTS CONDITION

The application of the Danckwerts condition on the enthalpy allows to express the normal conductive heat flux at the inlet boundary as proportional to the flow rate $\rho \mathbf{u}$ and the enthalpy variation ΔH between the upstream conditions and inlet conditions:

$$k \nabla T \cdot \mathbf{n} = \rho \Delta H \mathbf{u} \cdot \mathbf{n} \tag{4-21}$$

The enthalpy variation between the upstream conditions and inlet conditions, ΔH, depends in general both on the difference in temperature and in pressure, and is defined as:

$$\Delta H = \int_{T_{\mathrm{upstream}}}^{T} C_p dT + \int_{p_{\mathrm{upstream}}}^{p_{\mathrm{A}}} \frac{1}{\rho}(1 - \alpha_p T) dp \tag{4-22}$$

where T_{upstream} is the upstream temperature, T is the inlet temperature, p_{ustream} is the upstream absolute pressure, p_{A} is the inlet absolute pressure, C_p is the fluid heat capacity at constant pressure, ρ is the fluid density, and α_p is its coefficient of thermal expansion. See Equation 4-5 for details about the definition of the enthalpy.

In the unexpected case of a velocity field corresponding to an outgoing flow across the inlet boundary, a zero conductive flux condition is applied to avoid a unphysical conductive flux condition:

$$k \nabla T \cdot \mathbf{n} = 0, \qquad \mathbf{n} \cdot \mathbf{u} \geq 0$$

TEMPERATURE CONTRIBUTION TO THE INFLOW BOUNDARY CONDITION

The nature of the temperature contribution depends on the order of magnitude of advective and conductive heat transfer at the inlet.

For large flow rates, advective heat transfer dominates over conductive heat transfer at the inlet. In this case, the left-hand side of Equation 4-21 is small compared to its right-hand side. As the heat capacity C_p is positive, in the absence of pressure contribution to the enthalpy, this induces the following constraint on temperature

$$T - T_{\text{upstream}} = 0$$

which corresponds to a Dirichlet boundary condition that prescribes the upstream temperature at the inlet.

For low flow rates or in the presence of large heat sources or sinks next to the inlet, the conductive heat flux cannot be neglected. The first integral in Equation 4-22 has for effect to adjust the inlet temperature to balance the energy brought by the flow at the inlet and the energy transferred by conduction from the interior.

PRESSURE CONTRIBUTION TO THE INFLOW BOUNDARY CONDITION

Pressure losses in the virtual channel upstream to the inlet boundary are handled through the pressure contribution to the enthalpy (the second integral in Equation 4-22).

In addition to the cases where the upstream and inlet absolute pressures are equal, this term may be neglected when the work due to pressure changes is not included in the energy equation, or when the fluid is modeled as an ideal gas (in this case the coefficient of thermal expansion is the inverse of the temperature).

Theory for Heat Transfer in Moist Air

By making the ideal gas assumption in Equation 4-18, The Heat Transfer in Moist Air Interface solves the following equation:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot (\mathbf{q} + \mathbf{q_r}) = \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right) + \tau : \nabla \mathbf{u} + Q_H + Q \qquad (4\text{-}23)$$

involving the same variables as in Theory for Heat Transfer in Fluids, and the additional Q_H term, accounting for the diffusive flux of thermal enthalpy due to the rate of change of air and vapor in moist air. It is defined as (Ref. 5):

$$Q_H = -(C_{p,\,v} - C_{p,\,a}) \mathbf{g_w} \cdot \nabla T$$

where:

- $C_{p,v}$ is the specific heat capacity at constant pressure of vapor (SI unit: J/(kg·K))
- $C_{p,a}$ is the specific heat capacity at constant pressure of air (SI unit: J/(kg·K))
- $\mathbf{g_w}$ is the vapor flux by diffusion (SI unit: kg/(m^2·s))

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:

$$x_{\mathrm{vap}} = \frac{m_v}{m_a} = \frac{p_v M_v}{p_a M_a} \qquad (4\text{-}24)$$

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. 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:

$$\phi_w = \frac{p_v}{p_{sat}} \tag{4-25}$$

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, $p = p_v + p_a$ 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_w it has to be at the same temperature and pressure conditions. Then the thermodynamical properties of moist air can be deduced through the mixture formula described below.

The relative humidity 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_w reaches unity, it means that the vapor is saturated and that water vapor may condense.

SPECIFIC HUMIDITY

The *specific humidity* is defined as the ratio of water vapor, m_v, to the total mass, $m_{tot} = m_v + m_a$:

$$\omega = \frac{m_v}{m_{tot}} \tag{4-26}$$

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:

$$c_v = \frac{n_v}{V} \tag{4-27}$$

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:

$$c_{\text{sat}} = \frac{p_{\text{sat}}(T)}{RT} \tag{4-28}$$

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. 36, the saturation pressure can be defined using the following expression:

$$p_{\text{sat}}(T) = 610.7[\text{Pa}] \cdot 10^{7.5\frac{T - 273.15[\text{K}]}{T - 35.85[\text{K}]}} \tag{4-29}$$

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:

$$X_a = \frac{n_a}{n_{\text{tot}}} = \frac{p_a}{p} = \frac{p - \phi_w p_{\text{sat}}}{p} \tag{4-30}$$

$$X_v = \frac{n_v}{n_{\text{tot}}} = \frac{p_v}{p} = \frac{\phi_w p_{\text{sat}}}{p} \tag{4-31}$$

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_w is the relative humidity, and
- p_{sat} is the saturation pressure.

From Equation 4-30 and Equation 4-31, the following relation holds:

$$X_a + X_v = 1$$

Relative Humidity, Vapor Concentration, and Moisture Content
Moisture content and relative humidity can be related with the following expression:

$$\phi_w = \frac{c_v RT}{p_{sat}(T)} = \frac{x_{vap}p}{p_{sat}(T)\left(\dfrac{M_v}{M_a} + x_{vap}\right)} \tag{4-32}$$

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:

$$\rho_m = \frac{p}{RT}(M_a X_a + M_v X_v) \tag{4-33}$$

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. 37, the heat capacity at constant pressure of a mixture is:

$$C_{p, m} = \frac{M_a}{M_m} X_a C_{p, a} + \frac{M_v}{M_m} X_v C_{p, v} \tag{4-34}$$

where M_m represents the mixture molar fraction and is defined by

$$M_m = X_a M_a + X_v M_v$$

and where $C_{p, a}$ and $C_{p, v}$ are the heat capacity at constant pressure of dry air and steam, respectively.

Dynamic Viscosity

According to Ref. 37 and Ref. 38, the dynamic viscosity is defined as:

$$\mu_m = \sum_{i = a, v} \frac{X_i \mu_i}{\sum_j X_j \varphi_{ij}} \tag{4-35}$$

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. 38 and Ref. 37, the thermal conductivity of the mixture is defined similarly:

$$k_m = \sum_{i = a, v} \frac{X_i k_i}{\sum_j X_j \varphi_{ij}} \tag{4-36}$$

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 dry air properties are taken from the **Built-in** materials database, and the polynomials have been computed according

Ref. 39 for pure steam properties. The steam properties are based on the Industrial Formulation IAPWS-IF97.

The valid temperature range is $200 \text{ K} < T < 1200 \text{ K}$ for dry air properties and $273.15 \text{ K} < T < 873.15 \text{ K}$ 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.
- Saturation indicator satInd; this indicator is set to 1 if saturation has been detected ($\phi_w = 1$), 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 $0 \le \phi_w \le 1$, 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 4-25, Equation 4-28, and Equation 4-32:

$$c_v = \frac{x_{vap}p}{\left(x_{vap} + \dfrac{M_v}{M_a}\right)RT}$$

📄 | The concentration computation assumes that the ideal gas assumption is valid.

- ht.*feature*.fxvap(RH, T, pA), where RH is the relative humidity $0 \le \phi_w \le 1$, 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:

$$x_{\text{vap}} = \frac{\phi_w p_{\text{sat}}}{p - \phi_w p_{\text{sat}}} \cdot \frac{M_v}{M_a}$$

- `ht.`*feature*`.fpsat(T)`, where T is the temperature (SI unit: K). It returns the saturation pressure (SI unit: Pa) by using Equation 4-29.

- `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. 39, which provides steam properties based on the Industrial Formulation IAPWS-IF97. The temperature-dependency is as shown on Figure 4-1.
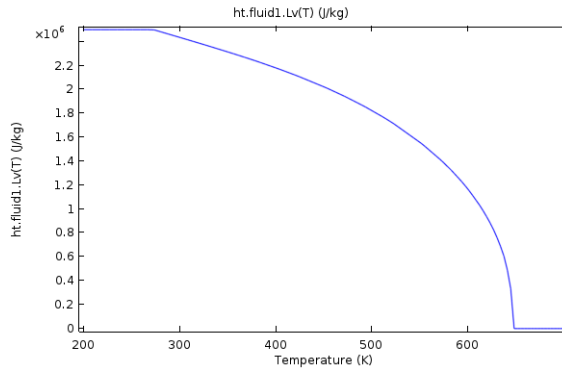
Figure 4-1: Latent heat of evaporation as a function of temperature.

Theory for Bioheat Transfer

The Bioheat Equation

The Bioheat Transfer Interface solves for the bioheat equation using Pennes' approximation (Ref. 6). 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:

$$\rho\, C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} \;=\; \rho_b\, C_{p,\,b}\, \omega_b (T_b - T) + Q_{\mathrm{met}} \qquad (4\text{-}37)$$

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: $\mathrm{kg/m^3}$)
- $C_{p,\,b}$ 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 Materials Library in the *COMSOL Multiphysics Reference Manual*.

Damaged Tissue

Add a Thermal Damage subnode under the Biological Tissue node to calculate tissue damage.

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

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

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

For the first two cases, the damaged tissue indicator, α, defined either by

$$\alpha = \frac{1}{t_{d, h}} \int_0^t \varphi_{d, h} dt$$

for hyperthermia analysis, or by

$$\alpha = \frac{1}{t_{d, c}} \int_0^t \varphi_{d, c} dt$$

for cryogenic analysis, with

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

is the ratio of the period of time when $T > T_{d, h}$ to the time limit $t_{d, h}$, or the ratio of the period of time when $T < T_{d, c}$ to the time limit $t_{d, c}$. 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 $\min(\alpha, 1)$.

For the last two cases, the necrosis time indicator, α_{necr}, defined either by

$$\alpha_{\text{necr}} = \int_0^t \varphi_{\text{n, h}} dt$$

for hyperthermia analysis, or by

$$\alpha_{\text{necr}} = \int_0^t \varphi_{\text{n, c}} dt$$

for cryogenic analysis, with

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

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

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

$$\theta_{\text{d}} = \begin{cases} 1 & \text{if } \alpha_{\text{necr}} > 0 \\ \min(\alpha, 1) & \text{otherwise} \end{cases} \tag{4-38}$$

Arrhenius Kinetics

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

$$\frac{\partial \alpha}{\partial t} = (1-\alpha)^n A e^{\frac{-\Delta E}{RT}}$$

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. 7) to be $A = 7.39 \cdot 10^{39}$ s^{-1} and $\Delta E = 2.577 \cdot 10^5$ J/mol. See Ref. 8, Ref. 9, and Ref. 10 for the characterization of these parameters for prostate, skin, and fat. See also Ref. 11 and Ref. 12 for more references on biological tissues material properties.

The fraction of necrotic tissue is then expressed by:

$$\theta_d = \min(\max(\alpha, 0), 1) \tag{4-39}$$

Thermal Properties

The material properties of the damaged tissue are redefined to take into account the influence of tissue injury. If ρ_d, $C_{p, d}$, 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, $k_{eff} = \theta_d k_d + (1 - \theta_d)k$
- The effective heat capacity at constant pressure, $(\rho C_p)_{eff} = \theta_d \rho_d C_{p, d} + (1 - \theta_d)\rho C_p$

In these equalities, θ_d takes one of the two definitions given above in Equation 4-38 or Equation 4-39 according to the integral form chosen.

Heat Source

A cooling or heating source is associated with the reaction leading to damage of tissue. Depending on the damage integral model, this source is expressed as follows:

- Temperature threshold for hyperthermia analysis:

$$Q = -\rho L_{d, h} \frac{\partial \theta_d}{\partial t}(T > T_{d, h})$$

- Temperature threshold for cryogenic analysis:

$$Q = \rho L_{d, c} \frac{\partial \theta_d}{\partial t}(T < T_{d, c})$$

- Energy absorption:

$$Q = -\rho L \frac{\partial \theta_d}{\partial t}$$

Theory for Heat Transfer in Porous Media

When considering heat transfer in a porous medium at the microscopic scale, two heat transfer equations can be established, for the solid and fluid phases. For undeformed immobile solids, Equation 4-16 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 4-18 becomes:

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

At the macroscopic scale, different modeling approaches are available, depending on the local thermal equilibrium hypothesis, and on the accuracy required on the thermal effects in the solid phase.

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

$$T_f = T_s = T \tag{4-40}$$

When this hypothesis can be assumed, 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. 13). This rule applies by multiplying the equation of the solid domain by the solid volume fraction, θ_s, multiplying the fluid equation by the porosity, ε_p, and summing resulting equations.

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

In the particular case of a packed bed of pellets, and under some assumptions about thermal conductivities ratio, the local thermal nonequilibrium model can be tuned to capture more precisely some thermal effects within the pellets. This is done by replacing the macroscale heat equation for the solid phase by an ordinary differential

equation for the microscale variation of temperature in the pellets along the radial coordinate. See the Packed Bed of Pellets section for more details.

When Should Local Thermal Equilibrium and Nonequilibrium 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 4-40. The Local Thermal Equilibrium section below details the derivation of the energy equation considering such assumption that remains accurately sufficient for several applications. Ref. 32 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. 33 provides criteria based on the dimensionless Sparrow number, $\mathbf{Sp}$, to indicate if temperature equilibrium is still valid or if a nonequilibrium point of view should be preferred. In Ref. 34, the influence of the Darcy number, $\mathbf{Da}$, and the ratio of phase conductivities is examined for transient heat transfer in packed beds. The Sparrow and Darcy numbers are defined by:

$$\mathbf{Sp} = \frac{h_{sf}L^2}{k_{eff}r_h} \qquad \mathbf{Da} = \frac{\kappa}{d^2}$$

where:

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

In the situations described in Ref. 33 and Ref. 34, small values of $\mathbf{Sp}$ (less than 100 or 500) and large values of $\mathbf{Da}$ (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 Nonequilibrium 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 framework for heat transfer in porous media where evaluating the validity of the equilibrium hypothesis is not required anymore.

Local Thermal Equilibrium

By making the local thermal equilibrium hypothesis for solid and fluid phases (Equation 4-40), The Heat Transfer in Porous Media Interface solves for the following version of the heat equation (Ref. 17), reformulated using a common temperature, T:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \rho_f C_{p,\,f} \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \qquad (4\text{-}41)$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \qquad (4\text{-}42)$$

The different quantities appearing here are:

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

$$(\rho C_p)_{\text{eff}} = \theta_s \rho_s C_{p,\,s} + \varepsilon_p \rho_f C_{p,\,f}$$

- ε_p is the porosity.
- θ_s is the solid matrix volume fraction.
- ρ_s is the solid matrix density.
- $C_{p,s}$ is the solid matrix heat capacity at constant pressure.
- 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 $\mathbf{u}_f = \mathbf{u}/\varepsilon_p$, where ε_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 4-41 disappear.

The effective thermal conductivity of the solid-fluid system, k_{eff}, is related to the conductivity of the solid, k_s, and to the conductivity of the fluid, k_f, and depends in a complex way on the geometry of the medium. In Ref. 13, Ref. 14, and Ref. 18, the following models are proposed:

- 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_f and k_s:

$$k_{eff} = \theta_s k_s + \varepsilon_p k_f$$

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_f and k_s:

$$\frac{1}{k_{eff}} = \frac{\theta_s}{k_s} + \frac{\varepsilon_p}{k_f}$$

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

- A good estimate is given by the weighted geometric mean of k_f and k_s, as long as k_f and k_s are not too different from each other

$$k_{eff} = k_s^{\theta_s} \cdot k_f^{\varepsilon_p}$$

- When the porous medium is composed of solid spherical inclusions in a fluid phase, the effective thermal conductivity can be expressed as:

$$k_{eff} = k_f \frac{2k_f + k_s - 2(k_f - k_s)\theta_s}{2k_f + k_s + (k_f - k_s)\theta_s}$$

- On the opposite, when the porous medium is composed of fluid spherical inclusions in a solid phase, the effective thermal conductivity can be expressed as:

$$k_{eff} = k_s \frac{2k_s + k_f - 2(k_s - k_f)\varepsilon_p}{2k_s + k_f + (k_s - k_f)\varepsilon_p}$$

- In the case of a wrapped screen, the effective thermal conductivity of the solid-fluid system can be defined as:

$$k_{eff} = k_f \frac{k_f + k_s - (k_f - k_s)\theta_s}{k_f + k_s + (k_f - k_s)\theta_s}$$

- Finally, when the porous medium is made of sintered metal fibers, the effective thermal conductivity of the solid-fluid system is defined as:

$$k_{eff} = \varepsilon_p^2 k_f + \theta_s^2 k_s + \frac{4\varepsilon_p \theta_s k_f k_s}{k_f + k_s}$$

Note that when k_f and k_s are equal, all these models give the same effective thermal conductivity. In addition, the effective conductivity is equal to k_f when the porosity is 1, and to k_s when the porosity is 0.

Local Thermal Nonequilibrium

The Local Thermal Nonequilibrium 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 NONEQUILIBRIUM HEAT TRANSFER

Nonequilibrium 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_s and $\varepsilon_p = 1 - \theta_s$ respectively, and with an additional source term quantifying exchanged heat between both phases (2.12 and 2.13 in Ref. 13):

$$\theta_s \rho_s C_{p,s} \frac{\partial T_s}{\partial t} + \nabla \cdot \mathbf{q}_s = q_{sf}(T_f - T_s) + \theta_s Q_s \tag{4-43}$$

$$\mathbf{q}_s = -\theta_s k_s \nabla T_s$$

$$\varepsilon_p \rho_f C_{p,f} \frac{\partial T_f}{\partial t} + \rho_f C_{p,f} \mathbf{u} \cdot \nabla T_f + \nabla \cdot \mathbf{q}_f = q_{sf}(T_s - T_f) + \varepsilon_p Q_f \tag{4-44}$$

$$\mathbf{q}_f = -\varepsilon_p k_f \nabla T_f$$

In these expressions:

- θ_s is the solid volume fraction (dimensionless)
- ε_p is the porosity (dimensionless)
- ρ_s and ρ_f are the solid and fluid densities (SI unit: kg/m³)
- $C_{p,\,s}$ 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/m²)
- 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/(m³·K))
- Q_s and Q_f are the solid and fluid heat sources (SI unit: W/m³)
- $\mathbf{u}$ is the fluid velocity vector (SI unit: m/s). 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 $\mathbf{u}_f = \mathbf{u}/\varepsilon_p$, where ε_p is the fluid's volume fraction, or equivalently the porosity.

The Porous Medium feature with **Porous medium type** set to **Local thermal equilibrium** 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. 35):

$$T = \frac{\theta_s \rho_s C_{p,\,s} T_s + \varepsilon_p \rho_f C_{p,\,f} T_f}{\theta_s \rho_s C_{p,\,s} + \varepsilon_p \rho_f C_{p,\,f}}$$

CORRELATION FOR THE INTERSTITIAL CONVECTIVE HEAT TRANSFER COEFFICIENT

The Porous Medium feature with **Porous medium type** set to **Local thermal equilibrium** provides a built-in correlation for q_{sf} for a packed bed of spherical pellets (2.14, 2.15, and 2.16 in Ref. 13):

$$q_{sf} = S_b h_{sf}$$

The specific surface area, S_b (SI unit: 1/m), for a bed packed with spherical particles of average diameter d_{pe} is:

$$S_b = \frac{6\theta_s}{d_{pe}}$$

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

$$\frac{1}{h_{sf}} = \frac{d_{pe}}{k_f \mathrm{Nu}} + \frac{d_{pe}}{\beta k_s}$$

where $\beta = 10$ for spherical particles, and Nu is the fluid-to-solid Nusselt number derived from following correlation (Ref. 15):

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

The Prandtl number, Pr, and particle Reynolds number, Re_p, are defined by:

$$\mathrm{Pr} = \frac{\mu C_{p,f}}{k_f} \qquad \mathrm{Re}_p = \frac{d_{pe} \rho_f \|\mathbf{u}\|}{\mu}$$

VOLUMETRIC AND SURFACE THERMAL CONDITIONS

Because each energy equation is multiplied by its volume fraction, θ_s and ε_p for solid and fluid phases respectively, a heat source or heat flux defined in one of the phases is also accounted with that ratio. As shown in Equation 4-43 and Equation 4-44, the volumetric heat sources $\theta_s Q_s$ and $\varepsilon_p Q_f$ are applied to the energy equations when Heat Source is added as a subnodes of the **Porous Matrix** or **Fluid** nodes to specify Q_s or Q_f.

Packed Bed of Pellets

For the modeling of packed beds of pellets, the The Heat Transfer in Packed Beds Interface interface implements an hybrid multiscale model of local thermal nonequilibrium.

As in the standard local thermal nonequilibrium model, two temperatures, one for the fluid phase and one for the pellets, are solved for. By assuming that the thermal conductivity of the pellets is much smaller than the one of the fluid phase, heat conduction among different pellets is neglected, and a 1D microscale equation for temperature conduction along the radial coordinate in the pellets is defined to replace the macroscale heat transfer equation. It is coupled to the macroscale heat transfer equation in the fluid phase, either by assuming temperature continuity, or a convective heat flux, at the outer surface of the pellets.

This model provides higher accuracy than the standard local thermal nonequilibrium model for highly nonlinear problems, in particular when combustion occurs within the pellets.

Spheres, cylinders, flakes, or any shape of pellets may be considered, as long as the specific surface area for heat exchange between the pellets and fluid phase can be determined. Porosity within the pellets themselves may be considered as well, in addition to the bed porosity.

EQUATIONS FOR HEAT TRANSFER IN A PACKED BED OF PELLETS

Macroscopic Heat Transfer in the Fluid

For the fluid phase, the heat equation is similar to the one solved in the standard local thermal nonequilibrium model:

$$\varepsilon_p \rho_f C_{p,\,f} \frac{\partial T_f}{\partial t} + \rho_f C_{p,\,f} \mathbf{u} \cdot \nabla T_f + \nabla \cdot \mathbf{q}_f = Q_{\mathrm{pe},\,f} + \varepsilon_p Q_f \qquad (4\text{-}45)$$

$$\mathbf{q}_f = -\varepsilon_p k_f \nabla T_f$$

In these expressions:

- ε_p is the packed bed porosity (dimensionless)
- ρ_f is the fluid density (SI unit: kg/m^3)
- $C_{p,\,f}$ is the fluid heat capacity at constant pressure (SI unit: J/(kg·K))
- $\mathbf{q}_f$ is the fluid conductive heat flux (SI unit: W/m^2)
- k_f is the fluid thermal conductivity (SI unit: W/(m·K))
- $Q_{\mathrm{pe},f}$ is the pellet-fluid heat exchange term (SI unit: $W/(m^3 \cdot K)$)
- Q_f is any fluid heat source (SI unit: W/m^3)
- $\mathbf{u}$ is the fluid velocity vector (SI unit: m/s). 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 $\mathbf{u}_f = \mathbf{u}/\varepsilon_p$, where ε_p is the fluid's volume fraction, or equivalently the porosity.

Microscopic Heat Transfer in the Pellets

In the pellets, the heat balance equation at normalized radius r is

$$(\rho C_p)_{\mathrm{pe,\,eff}} \frac{\partial T_{\mathrm{pe}}}{\partial t} + \frac{1}{(r r_{\mathrm{pe}})^2} \frac{\partial}{\partial r} \left(-r^2 k_{\mathrm{pe,\,eff}} \frac{\partial T_{\mathrm{pe}}}{\partial r} \right) = Q_{\mathrm{pe}} \qquad (4\text{-}46)$$

where

- $(\rho C_p)_{\mathrm{pe,eff}}$ is the effective volumetric heat capacity at constant pressure inside the pellet filled with fluid (SI unit: J/(kg·K)), defined by

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

- ρ_{pe} is the density of the porous matrix within the pellet (SI unit: kg/m^3)

- $C_{p,\text{pe}}$ is the volumetric heat capacity at constant pressure of the porous matrix within the pellet (SI unit: J/(kg·K))

- ε_{pe} is the porosity within the pellet (dimensionless)

- T_{pe} is the temperature along the radial coordinate of the pellet (SI unit: K)

- r_{pe} is outer radius of the pellet (SI unit: m)

- $k_{\text{pe,eff}}$ is the effective thermal conductivity of the pellet filled with fluid (SI unit: W/(m·K)), defined by

$$k_{\text{pe, eff}} = (1 - \varepsilon_{\text{pe}})k_{\text{pe}} + \varepsilon_{\text{pe}}k_{\text{f}}$$

- k_{pe} is the thermal conductivity of the porous matrix within the pellet (SI unit: W/(m·K))

- Q_{pe} is any heat source applied to the pellet (SI unit: W/m^3)

Coupling Condition at Pellet-Fluid Interface

As in the local thermal nonequilibrium model, the interstitial heat exchange between the fluid and the outer surface of the pellets is accounted for.

The heat flux at the outer surface of the pellet, applied in the microscale pellet equation, is

$$q_{\text{pe}|r = 1} = h_{\text{pe, f}}(T_{\text{f}} - T_{\text{pe}|r = 1})$$

with $h_{\text{pe,f}}$ the interstitial heat transfer coefficient (SI unit: W/(m^2·K)), defined as in the standard local thermal nonequilibrium model.

The corresponding heat exchange term applied in the macroscale heat transfer equation of the fluid is:

$$Q_{\text{pe, f}} = -S_{\text{b}}q_{\text{pe}|r = 1}$$

with S_{b} the specific surface area of the pellets. For a packed bed of spherical pellets, it is defined as

$$S_{\text{b}} = \frac{6(1 - \varepsilon_{\text{p}})}{d_{\text{pe}}}$$

with d_{pe} the diameter of the pellet (SI unit: m).

When the pellets are not spherical, a correction factor is applied to the radius, to extend this formula.

When continuity of temperatures at the fluid-pellet interface can be assumed, i.e. when

$$T_{\text{pe}|r=1} = T_{\text{f}}$$

then the interstitial heat flux can be rewritten as

$$q_{\text{pe}|r=1} = -\frac{1}{r_{\text{pe}}}\left(k_{\text{pe, eff}}\frac{\partial T_{\text{pe}}}{\partial r}\right)_{r=1}$$

Average Temperature in the Packed Bed

The porous temperature, T, has the following definition (Ref. 35):

$$T = \frac{(1-\varepsilon_{\text{p}})\dfrac{\displaystyle\int_{V_{\text{pe}}}(\rho C_p)_{\text{pe, eff}}T_{\text{pe}}dV}{V_{\text{pe}}} + \varepsilon_{\text{p}}\rho_{\text{f}}C_{p,\text{f}}T_{\text{f}}}{(1-\varepsilon_{\text{p}})\dfrac{\displaystyle\int_{V_{\text{pe}}}(\rho C_p)_{\text{pe, eff}}dV}{V_{\text{pe}}} + \varepsilon_{\text{p}}\rho_{\text{f}}C_{p,\text{f}}}$$

Theory for Heat Transfer in Moist Porous Media

A moist porous medium is filled with moist air and liquid water. By making the local thermal equilibrium hypothesis for all phases (solid matrix, moist air and liquid water), The Heat Transfer in Moist Porous Media Interface solves for the following heat equation, obtained from Equation 4-41 by accounting for the liquid water phase in the definition of the effective material properties and in the convective heat flux:

$$(\rho C_p)_{\text{eff}}\frac{\partial T}{\partial t} + (\rho_{\text{g}}C_{p,\text{g}}\mathbf{u}_{\text{g}} + \rho_{\text{l}}C_{p,\text{l}}\mathbf{u}_{\text{l}})\cdot\nabla T + \nabla\cdot\mathbf{q} = Q + Q_{\text{evap}} \qquad (4\text{-}47)$$

$$\mathbf{q} = -k_{\text{eff}}\nabla T \qquad (4\text{-}48)$$

The different quantities appearing here are:

- ρ_{g} and ρ_{l} (SI unit: kg/m^3) the moist air and liquid water densities.
- $C_{p,\text{g}}$ and $C_{p,\text{l}}$ (SI unit: J/(kg·K)) the moist air and liquid water heat capacities at constant pressure.

- $(\rho C_p)_{\mathrm{eff}}$ (SI unit: J/(m^3·K)) the effective volumetric heat capacity at constant pressure defined by an averaging model to account for the solid matrix, the moist air, and the liquid water properties:

$$(\rho C_p)_{\mathrm{eff}} = \theta_s \rho_s C_{p,\,s} + \varepsilon_p [(1 - s_1) \rho_g C_{p,\,g} + s_1 \rho_1 C_{p,\,1}]$$

- where s_1 is the liquid water saturation.
- $\mathbf{q}$ the conductive heat flux (SI unit: W/m^2).
- $\mathbf{u}_g$ (SI unit: m/s) the moist air velocity field, that should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area.
- $\mathbf{u}_1$ (SI unit: m/s) the liquid water velocity field, that should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area.
- k_{eff} (SI unit: W/(m·K)) the effective thermal conductivity (a scalar or a tensor if the thermal conductivity is anisotropic), defined by an averaging model to account for solid matrix, the moist air, and the liquid water properties. See Moist Porous Medium for details about the available averaging models.
- Q (SI unit: W/m^3), defined as the sum of the diffusive flux of enthalpy and of the liquid capillary flux:

$$Q = -[(C_{p,\,v} - C_{p,\,a})\mathbf{g}_w + C_{p,\,1}\mathbf{g}_{1c}]$$

See Theory for Moisture Transport in Porous Media for the definition of $\mathbf{g}_w$ and $\mathbf{g}_{1c}$.

- Q_{evap} (SI unit: W/m^3) the heat source (or sink) due to water phase change, defined as:

$$Q_{\mathrm{evap}} = L_v G_{\mathrm{evap}}$$

See Theory for Moisture Transport in Porous Media for the definition of G_{evap}.

Theory for Heat Transfer with Phase Change

Heat Transfer with Phase Change

Heat transfer and phase change are inter-related processes:

• Matter exists in a certain phase, liquid, solid or gas, at equilibrium under a given condition of pressure and temperature. The phase transition occurs when the temperature and pressure of the system come across a critical condition at which the material changes phase. The critical value of temperature at which the transition occurs for a given reference pressure can be computed through statistical physics or tabulated from experiments. In the following, we assume that the phase change is driven by the temperature, neglecting all other factors that may affect phase change, as equilibrium time, or purity of the material.

• At the macroscopic and mesoscopic level, phase change costs or releases a certain amount of energy which is called latent heat of phase transition, which is an input of the model. Because there is a balance between sensible and latent heat, phase change induces temperature variations.

The dedicated features for phase change simulation that are available in the Heat Transfer module handle models were it is assumed that the different phases are non miscible so that each phase is located in a distinct domain. The simulation predicts the position of the phase change interface and the associated thermal effects. The following sections present the two numerical methods available to do the modeling:

• Arbitrary Lagrangian-Eulerian Method
• Apparent Heat Capacity Method

Arbitrary Lagrangian-Eulerian Method

When the phase change interface is tracked using the arbitrary Lagrangian-Eulerian (ALE) method, the phase change interface velocity is defined explicitly by the Stefan condition, which expresses the energy balance at the interface.

It applies to the case of phase change between a solid (or immobile fluid) and a fluid phase, with a sharp transition between the phases, and no topology changes.

The Phase Change Interface boundary condition is applicable on the boundaries located at the interface between a solid and a fluid domain, two solid domains, two fluid domains or two porous medium domains.

The Phase Change Interface, Exterior boundary condition is applicable on the exterior boundaries adjacent to a solid, fluid, or porous medium domain.

The latent heat of phase change is taken into account through the definition of the phase change velocity.

The Stefan condition defines the phase change interface velocity v_n as follows:

$$v_n = \frac{\mathbf{q} \cdot \mathbf{n}_{mesh}}{\rho_s L_{s \to f}}$$

with $L_{s \to f}$ the latent heat of phase change from solid to fluid (SI unit: J/kg), ρ_s the solid density (SI unit: kg/m^3), and $\mathbf{q}$ the conductive heat flux jump across the interface (SI unit: W/m^2), defined as

$$\mathbf{q} = -k_s \nabla T_s + k_f \nabla T_f$$

with k_s the solid thermal conductivity (SI unit: W/(m·K)), k_f the fluid thermal conductivity (SI unit: W/(m·K)), and T_s, T_f the temperatures on each side of the interface.

When the density changes during the phase change, the volume change is compensated by a velocity at the interface in the fluid phase. The normal fluid velocity is expressed as:

$$v_f = \frac{\rho_f - \rho_s}{\rho_f} v_n$$

The phase change velocity v_n appearing in Stefan condition is relative to the solid position. In case of translation of the solid (using Translational Motion subfeature under Solid feature for instance), as in a continuous casting process, the solid translation velocity $\mathbf{u}_s$ contributes to v_n to describe the interface velocity relative to the spatial frame d$\mathbf{x}$/dt:

$$\frac{d\mathbf{x}}{dt} \cdot \mathbf{n} = v_n + \mathbf{u}_s \cdot \mathbf{n}$$

Similarly, the same term is added to the normal fluid velocity definition relative to the spatial frame:

$$v_f = \frac{\rho_f - \rho_s}{\rho_f} v_n + \mathbf{u}_s \cdot \mathbf{n}$$

Apparent Heat Capacity Method

The *apparent heat capacity* formulation provides an implicit capturing of the phase change interface, by solving for both phases a single heat transfer equation with effective material properties. The latent heat of phase change is taken into account by modifying the heat capacity.

It applies well to materials showing a mushy zone around the phase change interface. And it allows topology changes of the phase change interface.

The Phase Change Material domain condition should be used on both phases domains, to solve the heat equation after specifying the properties of the 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 $T_{pc} - \Delta T/2$ and $T_{pc} + \Delta T/2$. 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 $T_{pc} - \Delta T/2$ and to 0 after $T_{pc} + \Delta T/2$. The density, ρ, and the specific enthalpy, H, are expressed by:

$$\rho = \theta \rho_1 + (1 - \theta)\rho_2$$

$$H = \frac{1}{\rho}(\theta \rho_1 H_1 + (1 - \theta)\rho_2 H_2)$$

where the indices 1 and 2 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:

$$C_p = \frac{1}{\rho}(\theta_1 \rho_1 C_{p,1} + \theta_2 \rho_2 C_{p,2}) + (H_2 - H_1)\frac{d\alpha_m}{dT}$$

Here, θ_1 and θ_2 are equal to θ and $1-\theta$, respectively. See Figure 4-2 for details.
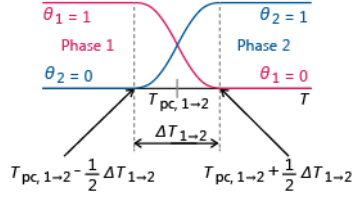
Figure 4-2: Phase indicators, phase change temperature, and transition interval.

The mass fraction, α_m, is defined from ρ_1, ρ_2 and θ according to:

$$\alpha_m = \frac{1}{2}\frac{\theta_2\rho_2 - \theta_1\rho_1}{\rho}$$

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}:

$$C_{eq} = \frac{1}{\rho}(\theta_1\rho_1 C_{p,\,1} + \theta_2\rho_2 C_{p,\,2})$$

and the distribution of latent heat C_L:

$$C_L(T) = (H_2 - H_1)\frac{d\alpha_m}{dT}$$

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

$$C_L(T) = L\frac{d\alpha_m}{dT}$$

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

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

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:

$$C_p = \frac{1}{\rho}(\theta_1\rho_1 C_{p,1} + \theta_2\rho_2 C_{p,2}) + C_L$$

and the ratio of specific heats, γ, is:

$$\gamma = \frac{C_p}{C_v} = \frac{\theta_1\rho_1 C_{p,1} + \theta_2\rho_2 C_{p,2}}{\theta_1\rho_1\dfrac{C_{p,1}}{\gamma_1} + \theta_2\rho_2\dfrac{C_{p,2}}{\gamma_2}}$$

The effective thermal conductivity reduces to:

$$k = \theta_1 k_1 + \theta_2 k_2$$

and the effective density is:

$$\rho = \theta_1\rho_1 + \theta_2\rho_2$$

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

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

PHASE CHANGE IN SOLID MATERIALS

When phase change is considered in solid materials, the density is defined on the material frame. Therefore a single density should be defined for the different phases to ensure mass conservation on the material frame:

$$\rho = \rho_1 = \rho_2$$

The expression of the specific enthalpy, H, simplifies to:

$$H = \theta H_1 + (1 - \theta)H_2$$

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

$$C_p = (\theta_1 C_{p,1} + \theta_2 C_{p,2}) + L \frac{d\alpha_m}{dT}$$

where the mass fraction is:

$$\alpha_m = \frac{\theta_2 - \theta_1}{2}$$

Theory for Heat Transfer in Building Materials

The Heat Transfer in Building Materials Interface solves for the following equations derived from Ref. 16:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = Q \tag{4-49}$$

$$\mathbf{q} = -(k_{\text{eff}} \nabla T + L_v \delta_p \nabla (\phi_w p_{\text{sat}})) \tag{4-50}$$

which is derived from Equation 4-15, considering the building material as a porous medium in local thermal equilibrium in which the following mixing rules apply:

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

$$(\rho C_p)_{\text{eff}} = \rho_s C_{p,s} + w(\phi_w) C_{p,w}$$

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, $w(\phi_w)$ (SI unit: kg/m^3) is the water content given by a moisture storage function, and $C_{p,w}$ (SI unit: $J/(kg \cdot K)$) is the water heat capacity at constant pressure.

- k_{eff} (SI unit: $W/(m \cdot K)$) is the effective thermal conductivity, defined as a function of the solid matrix and moisture properties:

$$k_{\text{eff}} = k_s \left(1 + \frac{b w(\phi_w)}{\rho_s}\right)$$

where k_s (SI unit: $W/(m \cdot K)$) 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:

$$L_v \delta_p \nabla (\phi_w p_{\text{sat}})$$

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

Theory for Harmonic Heat Transfer

When submitted to periodic sinusoidal heat loads at a given frequency, the temperature response of a body can, in some cases, be assumed periodic, sinusoidal, and of same frequency around an equilibrium temperature. If the temperature changes are small enough or if the thermal properties are constant, this time-dependent periodic problem may be replaced by an equivalent linear steady-state problem in the frequency domain, which is far less computationally expensive. The Heat Transfer interfaces support such frequency domain studies via the **Thermal Perturbation, Frequency Domain** study sequence.

In complex notations, the aforementioned temperature decomposition is expressed as:

$$T(t) \ = \ T_0 + T'e^{j\omega t} \tag{4-51}$$

where T_0 is the equilibrium temperature that verifies the steady-state heat transfer equation and may come from the solution of a previous study, T' is the complex amplitude of the harmonic perturbation around T_0, and ω is the angular frequency, related to the ordinary frequency, f, according to

$$\omega \ = \ 2\pi f$$

Note: The amplitude, T', is complex-valued since it includes the phase term $e^{j\varphi}$.

From the temperature decomposition in Equation 4-51, and according to the heat transfer equation in Equation 4-16, heat transfer by conduction in solids is then governed by:

$$j\omega\rho C_p T'e^{j\omega t} + \nabla \cdot (-k\nabla T_0) + \nabla \cdot (-k\nabla T')e^{j\omega t} \ = \ Q + Q'e^{j\omega t}$$

where $Q'e^{j\omega t}$ is the harmonic perturbation in domain around an average heat source, Q. Removing the terms of the steady-state heat transfer equation satisfied by T_0, and simplifying by $e^{j\omega t}$, this reduces to:

$$j\omega\rho C_p T' + \nabla \cdot (-k\nabla T') \ = \ Q'$$

which is the governing equation for T'. For constant material properties, the linearized form reads:

$$j\omega\rho_0 C_{p,0} T' + \nabla \cdot (-k_0 \nabla T') = Q' \qquad (4\text{-}52)$$

and for nonlinear material properties:

$$j\omega\rho_0 C_{p,0} T' + \nabla \cdot \left(- k_0 \nabla T' - T'\left(\frac{\partial k}{\partial T}\right)_{T_0} \nabla T_0\right) = Q' \qquad (4\text{-}53)$$

Here, ρ_0, $C_{p,0}$, and k_0 denote the density, heat capacity at constant pressure, and thermal conductivity, evaluated at T_0, that is: $\rho(T_0)$, $C_p(T_0)$, and $k(T_0)$, respectively.

When the linearized heat transfer equation, such as Equation 4-52 or Equation 4-53, can still describe the model accurately, the problem becomes steady-state in the frequency domain, therefore computationally less expensive than a time-dependent simulation. An automatic linearization process is performed by COMSOL Multiphysics so that no additional action is needed from the user to get these equations, even in the presence of temperature-dependent coefficients, in domains and boundaries. Only the expressions of the material properties and other parameters, as functions of the temperature, are required as for usual nonlinear modeling.

Harmonic loads can be prescribed through temperature harmonic constraints on boundaries or heat sources on domains and boundaries.

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 4-18 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:

$$m \overline{C_p} \frac{dT}{dt} + \int_S (\mathbf{n} \cdot \mathbf{q}) ds = \int_V Q dv \qquad (4\text{-}54)$$

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.
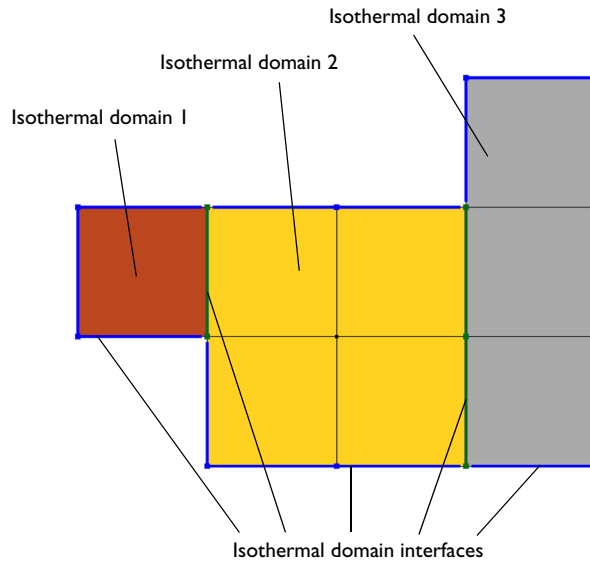
Figure 4-3: Interfaces at exterior boundaries of each Isothermal Domain.

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 $\varphi_{d \to u}$ or $\varphi_{u \to d}$, respectively, along or opposite to the geometrical normal vector. The Ventilation condition is written

$$-\mathbf{n}_d \cdot \mathbf{q}_d = \varphi_{d \to u} H_d - \varphi_{u \to d} H_u \tag{4-55}$$

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

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -h(T_u - T_d) \tag{4-56}$$

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

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -\frac{T_u - T_d}{R_{t,s}} \tag{4-57}$$

Theory for Heat Transfer in Thin Structures

In COMSOL Multiphysics, thin domains of solid, fluid or porous layered 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 Layered Materials
- Theoretical Background of the Different Formulations
- Thin Layer
- Thin Film
- Fracture
- Thin Rod

Modeling Layered Materials

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

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

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

Either the **Solid**, **Fluid**, or **Porous Medium** feature is available by default in each of these interfaces.

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) Solid (htlsh)	Thin Rod (ht)
Fluid	Fluid	Thin Film (ht) Fluid (htlsh)	—
Porous	Porous Medium	Fracture (ht) Porous Medium (htlsh)	—

All these functionalities have in common the fact that the thin domains they model are lumped into boundaries (for 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 4-4 shows an example where a thin structure significantly reduces the mesh density.

Copper wire modeled with a mesh

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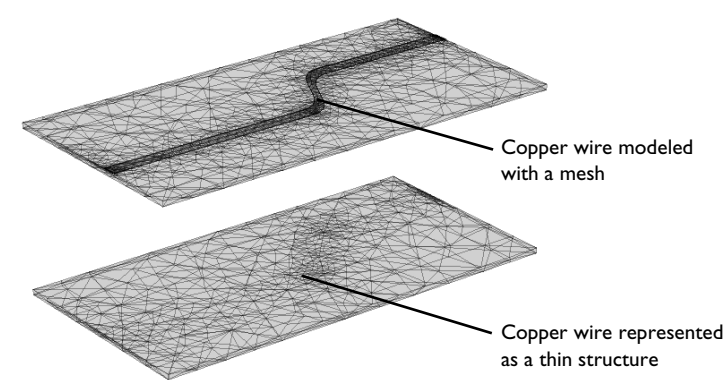
Figure 4-4: Modeling a copper wire as a domain (top) requires a denser mesh compared to modeling it as a boundary with a conductive layer (bottom).

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 **General** option of the Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface), Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface), or Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface) features.

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

Plotting and Evaluating Results in Layered Materials

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, $\mathbf{n}$, 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:

$$\nabla T = \nabla_{\mathbf{t}} T + \nabla_{\mathbf{n}} T \qquad\qquad (4\text{-}58)$$

This relation simplifies to:

$$\nabla T = \nabla_{\mathbf{t}} T \qquad \text{or} \qquad \nabla T = \nabla_{\mathbf{n}} T$$

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.

Equation 4-58 is valid for flat layered shells. For the curved ones, the gradient expression should account for the surface area and the scaling of each layer. The gradient in the product geometry of a curved layered shell with variable thickness can be written as:

$$\nabla T = \left[\left(\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r}\right)^{-1}\right]^{\mathrm{T}} \cdot (\nabla_{\mathbf{t}} T + \nabla_{\mathbf{n}} T)$$

with

$$\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} = \mathbf{I} + \left(z + \frac{d}{2}(-1 + z_{\mathrm{off}})\right)\nabla_{\mathbf{t}}\mathbf{n}$$

where

- $\mathbf{X}_r$ are the reference surface coordinates
- z is the extra-dimension thickness coordinate (varying between 0 and d)
- z_{off} is the relative midplane offset
- d is the layered material thickness

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 $\nabla_{\mathbf{t}}$.

In some applications, it is required to model variable thickness layers. This is achieved by scaling the constant thickness of the layer, d, using a thickness factor, lsc, which can be a function of the surface coordinates. The deformation gradient in a scaled product geometry of a curved layered shell can be written as

$$\nabla T = \left[\left(\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r}\right)^{-1}\right]^{\mathrm{T}} \cdot \left(\nabla_{\mathbf{t}} T + \frac{\nabla_{\mathbf{n}} T}{lsc}\right)$$

$$\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} = \mathbf{I} + \left(z + \frac{d}{2}(-1 + z_{\mathrm{off}})\right)\nabla_{\mathbf{t}}(lsc\,\mathbf{n})$$

AREA SCALE FACTOR

For a curved layered shell, the change in surface area of each layer should be accounted for while integrating the governing equations. The area scale factor for each layer of the layered material can be defined as

$$ASF = \left| \frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} \right|$$

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 Underlying 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:

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

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 4-15, and recalled here below:

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

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 4-58, the energy equation becomes

$$\rho\frac{\partial E_\mathrm{s}}{\partial t} + \rho\mathbf{u} \cdot (\nabla_\mathbf{t}E_\mathrm{s} + \nabla_\mathbf{n}E_\mathrm{s}) + \nabla \cdot (\mathbf{q} + \mathbf{q_r}) = -(\sigma{:}\mathbf{D}) + Q \qquad (4\text{-}59)$$

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 4-59 comes along with Fourier's law of conduction:

$$\mathbf{q} = -k(\nabla_\mathbf{t}T_\mathrm{s} + \nabla_\mathbf{n}T_\mathrm{s}) \qquad (4\text{-}60)$$

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

$$T_\mathrm{d} = (T_\mathrm{s})_{L\,=\,0} \qquad T_\mathrm{u} = (T_\mathrm{s})_{L\,=\,d_s} \tag{4-61}$$

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

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

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 4-15 becomes:

$$d_\mathrm{s}\rho\frac{\partial E}{\partial t} + d_\mathrm{s}\rho\mathbf{u}\cdot\nabla_{\boldsymbol{t}}E + \nabla_{\boldsymbol{t}}\cdot(\mathbf{q}_\mathrm{s} + \mathbf{q}_\mathrm{r}) = -d_\mathrm{s}(\sigma{:}\mathbf{D}) + d_\mathrm{s}Q + q_0 \tag{4-62}$$

$$\mathbf{q}_\mathrm{s} = -d_\mathrm{s}k\nabla_{\boldsymbol{t}}T \tag{4-63}$$

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 4-62 and Equation 4-63 have an additional factor, d_z, to account for the out-of-plane thickness.

When Equation 4-62 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:

$$-\mathbf{n}\cdot\mathbf{q} = d_\mathrm{s}Q_\mathrm{s} - d_\mathrm{s}\rho\frac{\partial E}{\partial t} - (d_\mathrm{s}\rho\mathbf{u}\cdot\nabla_{\boldsymbol{t}}E) - \nabla_{\boldsymbol{t}}\cdot(\mathbf{q}_\mathrm{s} + \mathbf{q}_\mathrm{r}) \tag{4-64}$$

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 4-15 becomes:

$$d_s \rho \frac{\partial E}{\partial t} + d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{n}} E + \nabla_{\mathbf{n}} \cdot (\mathbf{q}_s + \mathbf{q}_r) = -d_s(\sigma{:}\mathbf{D}) + d_s Q + q_0 \qquad (4\text{-}65)$$

$$\mathbf{q}_s = -d_s k \nabla_{\mathbf{n}} T \qquad (4\text{-}66)$$

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 4-65 and Equation 4-66 have an additional factor, d_z, to account for the out-of-plane thickness.

When Equation 4-65 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:

$$-\mathbf{n} \cdot \mathbf{q} = d_s Q_s - d_s \rho \frac{\partial E}{\partial t} - (d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{n}} E) - \nabla_{\mathbf{n}} \cdot (\mathbf{q}_s + \mathbf{q}_r) \qquad (4\text{-}67)$$

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 $(1/2)(T_u + T_d)$. The term $\nabla_\mathbf{n} \cdot (-d_s k \nabla_\mathbf{n} T)$ is then given by:

$$\nabla_\mathbf{n} \cdot (-d_s k \nabla_\mathbf{n} T) \approx -k_s \frac{T_d - 2T_{1/2} + T_u}{d_s}$$

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

$$-k_s \frac{T_u - T_d}{d_s} \qquad -k_s \frac{T_d - T_u}{d_s}$$

Similarly, the time-dependent term can be expressed using T_u and T_d by:

$$\rho C_p \frac{\partial T}{\partial t} \approx \rho C_p \frac{\partial T_{1/2}}{\partial t} = \frac{1}{2} \rho C_p \frac{\partial T_u}{\partial t} + \frac{1}{2} \rho C_p \frac{\partial T_d}{\partial t}$$

leading to:

$$-\mathbf{n}_d \cdot \mathbf{q}_d = \frac{d_s Q}{2} - \rho C_p \left(\frac{d_s}{2} \frac{\partial T_d}{\partial t} + \mathbf{u}_d \cdot \mathbf{n}_d T_d \right) - \left(-k_s \frac{T_u - T_d}{d_s} \right) - (\mathbf{q}_{r, d} \cdot \mathbf{n}_d) \quad \text{(4-68)}$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = \frac{d_s Q}{2} - \rho C_p \left(\frac{d_s}{2} \frac{\partial T_u}{\partial t} + \mathbf{u}_u \cdot \mathbf{n}_u T_u \right) - \left(-k_s \frac{T_d - T_u}{d_s} \right) - (\mathbf{q}_{r, u} \cdot \mathbf{n}_u) \quad \text{(4-69)}$$

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 (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) feature.

Figure 4-5 shows how Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) splits the temperature into T_u and T_d on interior boundaries:
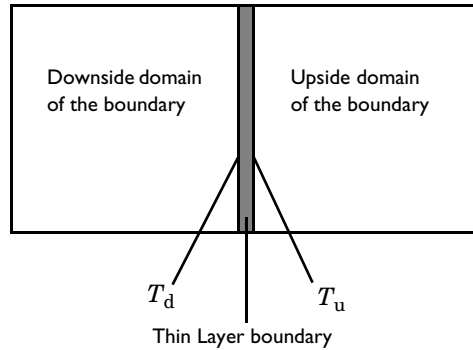
Figure 4-5: Upside and downside temperatures at a thin layer applied on an interior boundary. The thin layer is represented by the gray domain.

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:
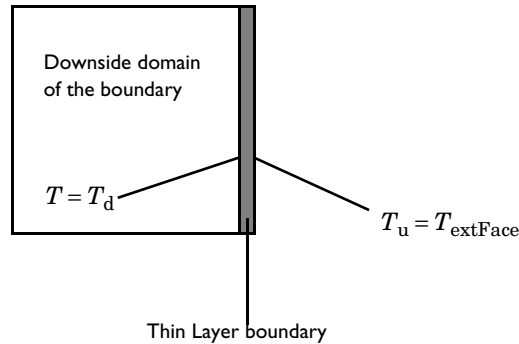
Figure 4-6: Upside and downside temperatures at a thin layer applied on an exterior boundary.

Table 4-3 summarizes the formulations available within the thin structure features of the Heat Transfer (ht) and Heat Transfer in Shells (htlsh) 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
Solid (htlsh)	Yes	Yes	No
Thin film (ht)	Yes	Yes	No
Fluid (htlsh)	Yes	Yes	No
Fracture (ht)	Yes	Yes	No
Porous Medium (htlsh)	Yes	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 4-59 to give:

$$\rho_{si} C_{p,\,si} \frac{\partial T_s}{\partial t} + \nabla_{\mathbf{t}} \cdot \mathbf{q}_{si} = Q_{si} \tag{4-70}$$

$$\mathbf{q}_{si} = -k_{si}(\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \tag{4-71}$$

$$T_d = (T_s)_{L\,=\,0}$$

$$T_u = (T_s)_{L\,=\,d_s}$$

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 (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) with **Layer type** set as **General** or more information about the boundary feature solving Equation 4-70 and Equation 4-71.

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 4-62 to Equation 4-64. Inside the thin layer, the heat equation becomes:

$$d_s \rho C_{p,s} \frac{\partial T}{\partial t} + \nabla_{\mathbf{t}} \cdot \mathbf{q}_s = d_s Q_s + q_0 \qquad (4\text{-}72)$$

$$\mathbf{q}_s = -d_s k \nabla_{\mathbf{t}} T \qquad (4\text{-}73)$$

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 4-72 and Equation 4-73 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 4-64, is received from the layer:

$$-\mathbf{n} \cdot \mathbf{q} = d_s Q_s - d_s \rho C_{p,s} \frac{\partial T}{\partial t} - \nabla_{\mathbf{t}} \cdot \mathbf{q}_s \qquad (4\text{-}74)$$

This formulation may be used to model efficiently a shell with multiple sandwiched layers with different material properties and thicknesses, by approximating the homogenized material properties as weighted averages of constant properties within each layer:

- The homogenized density is defined as

$$\rho = \frac{\sum_i \rho_i d_{s,i}}{\sum_i d_{s,i}}$$

- The homogenized heat capacity is defined as

$$C_p = \frac{\sum_i C_{p,i} d_{s,i}}{\sum_i d_{s,i}}$$

- The homogenized thermal conductivity is defined as

$$k = \frac{\sum_i (\mathbf{rot}_i \cdot \mathbf{A})^T \cdot k_i \cdot (\mathbf{rot}_i \cdot \mathbf{A}) d_{s,i}}{\sum_i d_{s,i}}$$

with $\mathbf{rot}_i$ the rotation tensor of layer i:

$$\mathbf{rot}_i = \begin{bmatrix} \cos(\theta) & \sin(\theta) & 0 \\ -\sin(\theta) & -\cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

and $\mathbf{A}$ the transformation matrix between the local and global coordinates systems:

$$\mathbf{A} = \begin{bmatrix} t_{1,x} & t_{2,x} & n_x \\ t_{1,y} & t_{2,y} & n_y \\ t_{1,z} & t_{2,z} & n_z \end{bmatrix}$$

See Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) with **Layer type** set as **Thermally thin approximation** for more information about the boundary feature solving Equation 4-74. See The Heat Transfer in Shells Interface for more information about the physics interface solving Equation 4-72.

- *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:

$$R_s = \frac{d_s}{k_s}$$

The heat flux across the thermally thick structure is derived from Equation 4-67 and gives

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -\frac{1}{2} d_s \rho_s C_{p,s} \frac{\partial T_d}{\partial t} - k_s \frac{T_u - T_d}{d_s} + \frac{1}{2} d_s Q_s \qquad (4\text{-}75)$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = -\frac{1}{2} d_s \rho_s C_{p,s} \frac{\partial T_u}{\partial t} - k_s \frac{T_d - T_u}{d_s} + \frac{1}{2} d_s Q_s \qquad (4\text{-}76)$$

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 4-77 and Equation 4-78:

$$d_{tot} = \sum_{j=1}^{n} d_{sj} \tag{4-77}$$

$$k_{tot} = \frac{d_{tot}}{\displaystyle\sum_{j=1}^{n} \frac{d_{sj}}{k_{sj}}} \tag{4-78}$$

where n is the number of layers.

> See Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) with **Layer type** set as **Thermally thick approximation** for more information about the boundary feature solving Equation 4-75 and Equation 4-76.

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 4-59 to give:

$$\rho C_p \frac{\partial T_s}{\partial t} + \rho C_p \mathbf{u} \cdot (\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) + \nabla_{\mathbf{t}} \cdot \mathbf{q}_f = Q_f \tag{4-79}$$

$$\mathbf{q}_f = -k(\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \tag{4-80}$$

$$T_d = (T_s)_{L=0}$$

$$T_u = (T_s)_{L=d_f}$$

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: kg/m^3)
- C_p is the heat capacity (SI unit: $J/(kg \cdot K)$)
- k is the thermal conductivity (SI unit: $W/(m \cdot K)$)
- Q_f is the heat source applied to the film (SI unit: W/m^3)
- d_f is the film 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 Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface) with **Thin film model** set as **General** for more information about the boundary feature solving Equation 4-79 and Equation 4-80.

THERMALLY THIN APPROXIMATION

The thermally thin approximation is derived from Equation 4-62 to Equation 4-64. Inside the thin layer, the heat equation becomes:

$$d_f \rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla_{\mathbf{t}} T \right) + \nabla_{\mathbf{t}} \cdot \mathbf{q}_f = d_f Q_f + q_0 \tag{4-81}$$

$$\mathbf{q}_f = -d_f k \nabla_{\mathbf{t}} T \tag{4-82}$$

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 4-72 and Equation 4-73 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 4-64, is received from the layer:

$$-\mathbf{n} \cdot \mathbf{q} = d_{\mathrm{f}}Q_{\mathrm{f}} - d_{\mathrm{f}}\rho C_p \frac{\partial T}{\partial t} - d_{\mathrm{f}}\rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T - \nabla_{\mathbf{t}} \cdot \mathbf{q}_{\mathrm{f}} \qquad (4\text{-}83)$$

> See Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface) with **Thin film model** set as **Thermally thin approximation** for more information about the boundary feature solving Equation 4-81. See The Heat Transfer in Films Interface for more information about the physics interface solving Equation 4-83.

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:

$$\mathbf{u} = \frac{\kappa}{\mu}\nabla_{\mathbf{t}}p$$

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 $\nabla_{\mathbf{t}}p$ 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 4-62 to Equation 4-64 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:

$$d_{\text{fr}}(\rho C_p)_{\text{eff}}\frac{\partial T}{\partial t} + d_{\text{fr}}\rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T + \nabla_{\mathbf{t}} \cdot \mathbf{q}_{\text{fr}} = d_{\text{fr}}Q + q_0 \qquad (4\text{-}84)$$

$$\mathbf{q}_{\text{fr}} = -d_{\text{fr}}k_{\text{eff}}\nabla_{\mathbf{t}} T \qquad (4\text{-}85)$$

Here $(\rho C_p)_{\text{eff}}$ 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, $\mathbf{q}_{\text{fr}}$ 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 4-64, is received from the fracture:

$$-\mathbf{n} \cdot \mathbf{q} = d_{\text{fr}}Q_0 - d_{\text{fr}}(\rho C_p)_{\text{eff}}\frac{\partial T}{\partial t} - d_{\text{fr}}\rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T - \nabla_{\mathbf{t}} \cdot \mathbf{q}_{\text{fr}} \qquad (4\text{-}86)$$

> See Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface) for more information about the boundary feature solving Equation 4-86. See The Heat Transfer in Fractures Interface for more information about the physics interface solving Equation 4-84.

Thin Rod

The Thin Rod feature is similar to Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) 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:

$$\int_{S(R)} Q\,ds = A_1 Q_1 - A_1 \rho_1 C_{p,1} + \frac{\partial T}{\partial t} - \nabla_{\mathbf{t}} \cdot \mathbf{q}_1 \qquad (4\text{-}87)$$

$$\mathbf{q}_1 = -A_1 k_1 \nabla_{\mathbf{t}} T \qquad (4\text{-}88)$$

with

$$A_1 = \pi r_1^2$$

	See Thin Rod for node information.

Theory for the Lumped Thermal System Interface

The Lumped Thermal System Interface uses a discrete network representation of thermal systems to model heat transfer by analogy with electrical circuits. The domain and boundary conditions for heat transfer are idealized by components joined by a network of perfectly thermally conductive wires.

The analogy with electrical circuits is made by considering the temperature difference across the network components as the effort variable, and the heat rate through the network component as the flow variable.

By solving for heat balance within the circuit, the temperatures at the nodes in the circuit can be determined.

This section presents the underlying theory of the Lumped Thermal System interface, and the resulting heat transfer equations that hold.

In this section:

- Temperature Differences and Heat Rates in the Lumped Thermal System
- Theory for the Conductive Thermal Resistor Component
- Theory for the Radiative Thermal Resistor Component
- Theory for the Thermal Capacitor and Thermal Mass Components
- Theory for the Heat Pipe Component
- Theory for the Thermoelectric Module Component

Temperature Differences and Heat Rates in the Lumped Thermal System

The Lumped Thermal System interface uses components to idealize heat transfer in domains and on boundaries.

For each two-port component, the temperature difference across the component, ΔT, is expressed as:

$$\Delta T = T_{p2} - T_{p1}$$

Note that the temperature difference is attached to a component, and has therefore the same value at its two ports p1 and p2. It is available for postprocessing, for example with the lts.R1_DT variable, when considering a **Conductive Thermal Resistor** component with name R1.

The heat rate through a two-port component, P, is expressed in different ways depending on the represented device. Then, the heat rates at ports p1 and p2, P_{p1} and P_{p2}, are defined from P. The different cases are summarized in Table 4-4.

TABLE 4-4: EXPRESSIONS OF THE TWO-PORT COMPONENTS HEAT RATES IN THE LUMPED THERMAL SYSTEM INTERFACE

	HEAT RATE THROUGH THE COMPONENT	HEAT RATE AT PORT P1	HEAT RATE AT PORT P2
Name	P	P_{p1}	P_{p2}
Expression in thermal resistors and heat pipe			
Conductive Thermal Resistor	$-\dfrac{\Delta T}{R}$	$-P$	P
Convective Thermal Resistor	$-\dfrac{\Delta T}{R_{\text{conv}}}$	$-P$	P
Radiative Thermal Resistor	$-\dfrac{\Delta T}{R_{\text{rad}}}$	$-P$	P
Heat Pipe	$-\dfrac{\Delta T}{R}$	$-P$	P
Expression in the other two-port components			
Thermal Capacitor	$-C\dfrac{\partial \Delta T}{\partial t}$	$-P$	P
Heat Rate Source	P_{src}	0	P
Thermoelectric Module	$P_{1,\text{src}} + P_{2,\text{src}}$	$P_{1,\text{src}}$	$P_{2,\text{src}}$

Note that for thermal resistors components, the heat rate at port p1 has the same sign as the temperature difference ΔT across the component. In practice, when the temperature at port p2 is higher than the temperature at port p1 ($\Delta T > 0$), applying conductive heat transfer between these two ports is equivalent to apply a heat source (positive heat rate P_{p1}) at port p1.

For the heat pipe component, the port p1 should correspond to the evaporator side (hot side), and the port p2 should correspond to the condenser side (cold side). Therefore, the temperature difference ΔT across the component should be negative in

normal operating conditions. With this setting, P_{p1} is always negative as well, meaning that cooling is applied on the hot side.

For the heat rate source component, the heat rate is fully applied on port p2.

For the thermoelectric module component, the heat rates $P_{1,src}$ and $P_{2,src}$ can be expressed in different ways. See Theory for the Thermoelectric Module Component for more details.

Theory for the Conductive Thermal Resistor Component

This section presents the underlying theory of the Conductive Thermal Resistor two-port component.

THERMAL RESISTANCE ANALOGY FOR STEADY CONDUCTION

By analogy with Ohm's law for electric current, the heat rate P (SI unit: W) in a material due to the temperature difference ΔT (SI unit: K) is

$$P = -\frac{\Delta T}{R} \tag{4-89}$$

where R (SI unit: K/W) is the thermal resistance.

The expression of the thermal resistance used in Equation 4-89 depends on the geometric configuration.

Plane Shell
When considering steady conduction through a plane shell of surface area A (SI unit: m²), thickness L (SI unit: m), and constant thermal conductivity k (SI unit: W/(m·K)), the thermal resistance R is:

$$R = \frac{L}{kA}$$

Cylindrical Shell
When considering steady conduction through a cylindrical shell of inner radius r_i and outer radius r_o (SI unit: m), height H (SI unit: m), and constant thermal conductivity k (SI unit: W/(m·K)), the thermal resistance R is:

$$R = \frac{1}{2\pi kH}\ln\left(\frac{r_o}{r_i}\right)$$

Spherical Shell

When considering steady conduction through a spherical shell of inner radius r_i and outer radius r_o (SI unit: m), and constant thermal conductivity k (SI unit: W/(m·K)), the thermal resistance R is:

$$R = \frac{1}{4\pi k}\left(\frac{1}{r_i} - \frac{1}{r_o}\right)$$

STEADY CONDUCTION THROUGH A COMPOSITE WALL

When considering several materials, Equation 4-89 can be updated to get a relationship between the heat transfer rate P and the overall temperature difference $\Delta T_{overall}$ (SI unit: K):

$$P = -\frac{\Delta T_{overall}}{R_{tot}}$$

where R_{tot} (SI unit: K/W) is the total thermal resistance.

Thermal Resistances in Series

When considering steady conduction through a composite wall made of n layers of thermal resistances $R_1,...,R_n$, see Figure 4-7, the total resistance is defined by:

$$R_{tot} = \sum_{i=1}^{n} R_i$$

with

$$R_i = \frac{L_i}{k_i A}$$
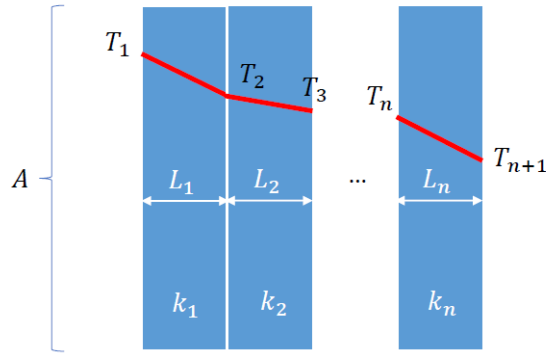
Figure 4-7: Steady conduction through a composite wall made of n layers

This corresponds to the following serial thermal circuit representation:
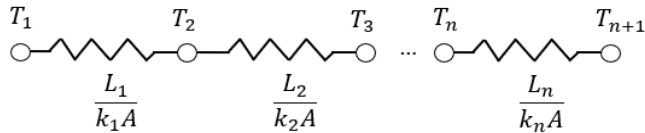
where $\Delta T_{\text{overall}} = T_{n+1} - T_1$. Note that by analogy with electrical circuits, the elements placed in series in the circuit share the heat rate (flow variable).

Thermal Resistances in Parallel

When considering steady conduction through a composite wall made of n slabs of thermal resistances $R_1,...,R_n$, see Figure 4-8, the total resistance is defined by:

$$R_{\text{tot}} = \left(\sum_{i=1}^{n} \frac{1}{R_i} \right)^{-1}$$

with

$$R_i = \frac{L}{k_i A_i}$$
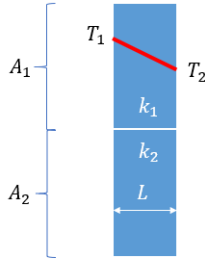
Figure 4-8: Steady conduction through a composite wall made of n slabs.

This corresponds to the following parallel thermal circuit representation:
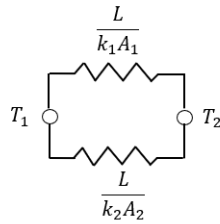
where $\Delta T_{overall} = T_2 - T_1$. Note that by analogy with electrical circuits, the elements placed in parallel in the circuit share the temperature difference (effort variable).

CONVECTIVELY ENHANCED CONDUCTIVITY

See Equivalent Thermal Conductivity Correlations for details about the correlations used to modify the thermal conductivity to account for convective heat flux.

RADIATION IN OPTICALLY THICK PARTICIPATING MEDIUM

See Rosseland Approximation Theory for details about how to modify the thermal conductivity to account for radiative heat flux when the optical thickness of the medium is large.

Theory for the Radiative Thermal Resistor Component

Let's consider a set of opaque diffuse gray surfaces. The net radiative heat rate from the surface of index i is expressed in function of its radiosity and irradiation:

$$P_i = \frac{E_{bi} - J_i}{R_i}$$

where the surface radiative resistance R_i is defined as:

$$R_i = \frac{1 - \varepsilon_i}{A_i \varepsilon_i}$$

with A_i the surface area (SI unit: m²) and ε_i the surface emissivity (dimensionless). See Theory for Surface-to-Surface Radiation for more details.

This allows a network representation of the radiative heat rate with E_{bi} and J_i as nodes. The surface resistance tends toward 0 for a large surface or a surface with large emissivity, for which the blackbody approximation holds.

When considering the surface of index i as part of an enclosure, the net radiative heat rate is expressed as:

$$P_i = \sum_{j=1}^{n} A_i F_{ij}(J_i - J_j) = \sum_{j=1}^{n} \frac{E_{bi}(J_i - J_j)}{R_{ij}}$$

where the space resistance is

$$R_{ij} = \frac{1}{A_i F_{ij}}$$

where F_{ij} is the view factor for surfaces i and j.

TWO-SURFACE ENCLOSURE

In the network representation of the radiative heat transfer in a two-surface enclosure, the surface resistance and the space resistance are connected in a serial way. Therefore the total radiative resistance is expressed as follows:

$$R_{12} = \frac{1 - \varepsilon_1}{A_1 \varepsilon_1} + \frac{1}{A_1 F_{12}} + \frac{1 - \varepsilon_2}{A_2 \varepsilon_2}$$

And the net radiation exchange is expressed as:

$$P_{12} = \frac{\sigma(T_1^4 - T_2^4)}{R_{12}}$$

For specific configurations, the surface areas can be evaluated from the geometric dimensions and the view factors are equal to 1. Therefore the expression above simplifies (see Ref. 21 for details).

Concentric Spheres
The surface areas A_1 and A_2 verify

$$\frac{A_1}{A_2} = \left(\frac{r_1}{r_2}\right)^2$$

And the net radiation exchange is

$$P_{12} = \frac{A_1\sigma(T_1^4 - T_2^4)}{\dfrac{1}{\varepsilon_1} + \dfrac{1-\varepsilon_2}{\varepsilon_2}\left(\dfrac{r_1}{r_2}\right)^2}$$

Large (infinite) Parallel Planes
The surface areas A_1 and A_2 verify

$$A_1 = A_2$$

And the net radiation exchange is

$$P_{12} = \frac{A_1\sigma(T_1^4 - T_2^4)}{\dfrac{1}{\varepsilon_1} + \dfrac{1}{\varepsilon_2} - 1}$$

Long (infinite) Concentric Cylinders
The surface areas A_1 and A_2 verify

$$\frac{A_1}{A_2} = \frac{r_1}{r_2}$$

And the net radiation exchange is

$$P_{12} = \frac{A_1\sigma(T_1^4 - T_2^4)}{\dfrac{1}{\varepsilon_1} + \dfrac{1-\varepsilon_2}{\varepsilon_2}\left(\dfrac{r_1}{r_2}\right)}$$

These expressions are predefined options in the Radiative Thermal Resistor feature. Further simplifications can be made on the net radiation exchange when the blackbody assumption holds ($\varepsilon_1 = \varepsilon_2 = 1$).

Theory for the Thermal Capacitor and Thermal Mass Components

This section presents the underlying theory of the Thermal Capacitor two-port component and the Thermal Mass one-port component.

THERMAL CAPACITANCE

The thermal capacitance C (SI unit: J/K) is a measure of how much heat a body can store. It is defined as:

$$C = V\rho C_p = mC_p$$

with V the volume (SI unit: m^3), ρ the density (SI unit: kg/m^3), C_p the heat capacity at constant volume (SI unit: J/(kg·K)), and m the mass (SI unit: kg).

LUMPED THERMAL CAPACITANCE MODEL

Under certain conditions, the temperature T of a solid body subjected to convective heat transfer on its boundaries may be considered as homogeneous, depending only on time, using the thermal capacitance:

$$\frac{T - T_\infty}{T_0 - T_\infty} = e^{-\frac{hA}{C}t}$$

where:

- T_0 is the initial body temperature
- T_∞ is the surrounding fluid temperature
- h is the convective heat transfer coefficient (SI unit: W/(m²·K))
- A is the surface of convective heat transfer (SI unit: m²)

This approximation, referred as the lumped thermal capacitance model (see Ref. 41), holds when h, C, and A are constant and the gradients of temperature within the body are expected to be smaller than the gradients of temperature between the body and the surrounding. It happens for example when the thermal contact between the solid and the fluid is poor, or when the solid is a good thermal conductor.

This is characterized by the Biot number Bi (dimensionless):

$$\text{Bi} = \frac{hL}{k}$$

where L is a relevant length scale of the body, and k is its thermal conductivity.

The lumped thermal capacitance model is usually assumed to be valid when $\text{Bi} < 0.1$.

Regarding the network representation of thermal systems, the Biot number may be used to determine how many nodes should be included, assuming an homogeneous temperature distribution in the corresponding domains.

Theory for the Heat Pipe Component

Heat pipes are devices which use the latent heat of evaporation and condensation of a fluid to transfer heat from a source to a sink with high efficiency.

DEVICE DESCRIPTION

Although various geometric configurations are available, a heat pipe includes a vapor channel delimited by a solid wall, with a porous wick in between, see Figure 4-9 below.
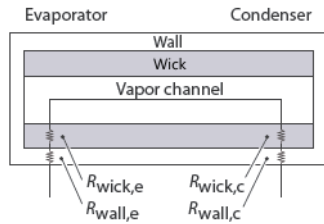
Figure 4-9: Heat pipe components.

The working fluid flows from the hot side to the cold side of the heat pipe under vapor state in the core channel, and under liquid state on the way back through the porous wick by capillary action. It evaporates when leaving the wick to the core channel on the hot side, named evaporator side, and condensates when entering the wick on the cold side, named condenser side.

The heat is transferred by conduction through the wall, and by conduction and convection in the wick and in the vapor channel. The latent heat absorbed and released by the evaporation and condensation of the working fluid makes heat pipes very effective heat transfer devices, with large effective thermal conductivities.

NETWORK REPRESENTATION

Whereas the full modeling of heat transfer in a heat pipe necessitates the modeling of several processes (vapor transport in the channel, liquid transport in the porous wick, and phase change at the interface between the wick and the vapor channel), a network representation allows to estimate the temperature drop between the evaporator and the condenser sides by considering a set of relevant thermal resistances.

By making the analogy with electrical circuits, the total thermal resistance R (SI unit: W/K) is expressed as:

$$R = R_{\text{wall, e}} + R_{\text{wick, e}} + R_{\text{lv, e}} + R_{\text{a}} + R_{\text{lv, c}} + R_{\text{wick, c}} + R_{\text{wall, c}}$$

where

$$R_{\text{a}} = \cfrac{1}{\cfrac{1}{R_{\text{v, a}}} + \cfrac{1}{R_{\text{wick, a}}} + \cfrac{1}{R_{\text{wall, a}}}}$$

with

- $R_{\text{wall,e}}$ the thermal resistance of the wall on evaporator side
- $R_{\text{wick,e}}$ the thermal resistance of the wick on evaporator side
- $R_{\text{lv,e}}$ the thermal resistance of the liquid-vapor interface on evaporator side
- $R_{\text{lv,c}}$ the thermal resistance of the liquid-vapor interface on condenser side
- $R_{\text{wick,c}}$ the thermal resistance of the wick on condenser side
- $R_{\text{wall,c}}$ the thermal resistance of the wall on condenser side
- $R_{\text{v,a}}$ the thermal resistance of the vapor channel in adiabatic section
- $R_{\text{wick,a}}$ the thermal resistance of the wick in adiabatic section
- $R_{\text{wall,a}}$ the thermal resistance of the wall in adiabatic section

By making assumptions on the relative magnitude of the thermal resistances, the expression of R reduces to:

$$R = R_{\text{wall, e}} + R_{\text{wick, e}} + R_{\text{wick, c}} + R_{\text{wall, c}}$$

And the temperature drop $\Delta T = T_{\text{condenser}} - T_{\text{evaporator}}$ is then expressed as

$$\Delta T = -\frac{P}{R}$$

where P (SI unit: W) is the operating power of the heat pipe.

This corresponds to the following network representation:
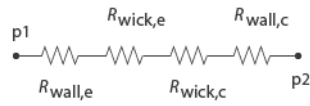
Figure 4-10: Thermal network representation of a heat pipe.

See Ref. 18 for details about the network representation of heat pipes and the expressions of effective thermal conductivity of the wick used to express its thermal resistance.

Theory for the Thermoelectric Module Component

Thermoelectric modules are devices which use thermoelectricity (Peltier effect) to transfer heat from a source to a sink.

DEVICE DESCRIPTION

They are composed of thermocouples, each consisting of p-type and n-type semiconductors, connected electrically in series and thermally in parallel and sandwiched between two high thermally conductive but low electrically conductive ceramic plates, as described on Figure 4-11 below.
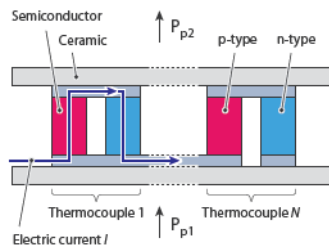
Figure 4-11: Thermoelectric module schematic description.

See Theory for the Thermoelectric Effect Interface for details about the Peltier effect.

HEAT RATE

In heating and cooling operating modes of the thermoelectric module (that is, no heat generation), the heat rates at the cold and hot junctions of each thermocouple are due to Peltier effect, Joule heating, thermal conduction, and heat storage:

- Hot junction:

$$P_{\text{hot}} = -SIT_{\text{hot}} + \frac{R_e I^2}{2} + \frac{\Delta T}{R} - C_{\text{hot}} \frac{\partial T_{\text{hot}}}{\partial t} \tag{4-90}$$

- Cold junction:

$$P_{\text{cold}} = SIT_{\text{cold}} + \frac{R_e I^2}{2} - \frac{\Delta T}{R} - C_{\text{cold}} \frac{\partial T_{\text{cold}}}{\partial t} \tag{4-91}$$

where

- S (SI unit: V/K) is the Seebeck coefficient of the thermocouple
- I (SI unit: A) is the electric current operating in the module
- R_e (SI unit: Ω) is the electrical resistance of the thermocouple
- ΔT (SI unit: K) is the temperature difference between the two sides of the module
- R (SI unit: K/W) is the thermal resistance of the thermocouple
- C_{hot} and C_{cold} (SI unit: J/K) are the thermal capacitances on each side of the module

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

The Peltier effect is a cooling effect at the hot junction, and a heating effect at the cold junction.

In Equation 4-90 and Equation 4-91 the assumption of a symmetric distribution of Joule heating at the hot and cold junctions has been made.

THERMOCOUPLE ELECTRICAL AND THERMAL RESISTANCES

As the p-type and n-type semiconductors are connected electrically in series, the electrical resistance R_e of each thermocouple is expressed as:

$$R_e = R_{e,\,p} + R_{e,\,n}$$

where $R_{e,p}$ and $R_{e,n}$ (SI unit: Ω) are the electrical resistances of the p-type and n-type semiconductors.

And as they are connected thermally in parallel, the thermal resistance R of each thermocouple is expressed as:

$$\frac{1}{R} = \frac{1}{R_p} + \frac{1}{R_n}$$

where R_p and R_n (SI unit: K/W) are the thermal resistances of the p-type and n-type semiconductors.

THERMOCOUPLE SEEBECK COEFFICIENT

Finally, the Seebeck coefficient S of each thermocouple is expressed as:

$$S = S_p - S_n$$

TOTAL HEAT RATE

The total heat rate through the module is obtained by summing the heat rates in each thermocouples.

NETWORK REPRESENTATION

The following network representation corresponds to Equation 4-90 and Equation 4-91:
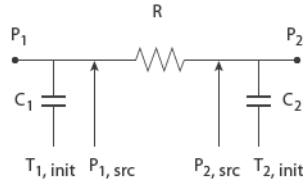
Figure 4-12: Thermal network representation of a thermoelectric module.

See Ref. 18 and Ref. 19 for details about the network representation of thermoelectric modules.

PERFORMANCE GRAPHS FOR THERMOELECTRIC COOLERS

It is usual to characterize thermoelectric coolers through graphs showing the heat removed from the cold side of the module, Q, in function of the difference of temperature ΔT between the hot and cold sides, the input intensity I, and the ambient temperature.

Linearized Model at Given Intensity and Ambient Temperature

This model establishes a linear relation between the removed heat and the temperature difference, as shown on the figure below:
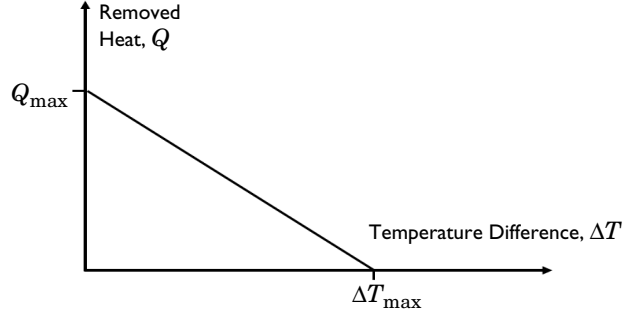
Q_{max} is the maximum amount of heat that the thermoelectric cooler can remove, when there is no temperature difference between the two sides of the module.

The linear curves from ($\Delta T=0$, $Q=Q_{max}$) to ($\Delta T=\Delta T_{max}$, $Q=0$) define the removed heat Q as:

$$Q = Q_{max} - \left(Q_{max} \cdot \frac{\Delta T}{\Delta T_{max}} \right)$$

Note that this relation is valid when the temperature difference satisfies:

$$0 \le \Delta T \le \Delta T_{max}$$

Then the powers applied at the cold and hot sides are respectively:

- P=-Q (removed heat)
- P=Q + $R_e I^2$ (waste heat, including Joule heating)

For cases where the current through the device varies, the performance parameters should be made a function of the current.

General Model

The removed heat may be defined as a more general function of the temperature difference and other parameters such as the temperature at the hot side.

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 for Opaque Surfaces
- Deriving the Radiative Heat Flux for Semitransparent Surfaces
- Wavelength Dependence of Surface Properties
- Directional Dependence of Surface Properties
- The Radiosity Method for Diffuse-Gray Surfaces
- The Radiosity Method for Diffuse-Spectral Surfaces
- View Factor Evaluation

Deriving the Radiative Heat Flux for Opaque Surfaces

In Figure 4-13, consider a point P located on a surface that has an emissivity ε, diffuse reflectivity ρ_d, specular reflectivity ρ_s, 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.
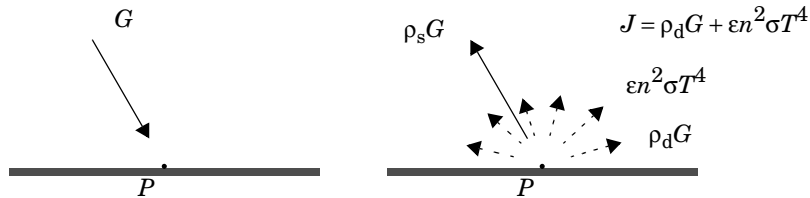

Figure 4-13: Incoming irradiation (left), outgoing radiosity (right).

The total incoming radiative flux at P is called *irradiation* and denoted G. The total diffuse outgoing radiative flux at P is called *radiosity* and denoted J. This radiosity is the sum of diffusely reflected and emitted radiation:

$$J = \rho_{d}G + \varepsilon e_{b}(T) \tag{4-92}$$

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 outgoing radiation (radiosity and specular reflected radiation):

$$q = G - (J + \rho_{s}G)$$

which can be also written as

$$q = (1 - \rho_{s})G - J \tag{4-93}$$

Using Equation 4-92 and Equation 4-93, 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.

$$q = (1 - (\rho_{d} + \rho_{s}))G - \varepsilon e_{b}(T) \tag{4-94}$$

Most opaque bodies also behave as ideal gray bodies, meaning that the absorptivity and emissivity are equal, and the reflectivity $\rho_{d} + \rho_{s}$ is therefore obtained from the following relation:

$$\alpha = \varepsilon = 1 - (\rho_{d} + \rho_{s}) \tag{4-95}$$

Thus, for ideal gray bodies, q is given by:

$$q = \varepsilon(G - e_{b}(T)) \tag{4-96}$$

This is the expression used for the radiative boundary condition.

Deriving the Radiative Heat Flux for Semitransparent Surfaces

In Figure 4-13, consider a point P located on a semitransparent surface that has an emissivity ε_{u}, diffuse reflectivity $\rho_{d,u}$, specular reflectivity $\rho_{s,u}$, refractive index n_{u}, and temperature T_{u} on the upside, and an emissivity ε_{d}, diffuse reflectivity $\rho_{d,d}$, specular

reflectivity $\rho_{s,d}$, refractive index n_d, and temperature T_d on the downside. As the surface is assumed semitransparent, some radiation is transmitted through the body.

$$J_u = \rho_{d,u}G_u + \varepsilon_u n_u^2 \sigma T^4$$
$$\varepsilon_u n_u^2 \sigma T^4$$
$$\rho_{d,u}G_u$$

G_u

$\rho_{s,u}G_u$

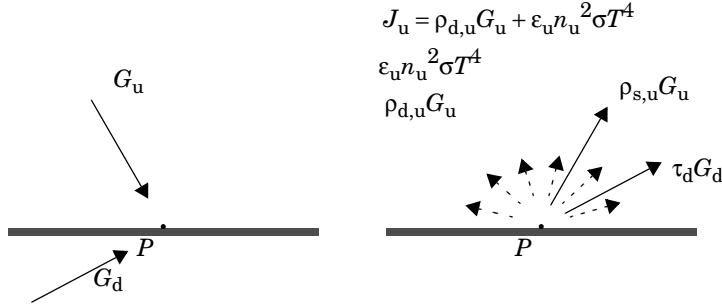$\tau_d G_d$

P

G_d

P

Figure 4-14: Upside and downside incoming irradiation (left), upside outgoing radiosity (right). The downside outgoing radiosity is defined in a similar way.

The total incoming radiative flux at P is called *irradiation*, and is denoted G_u on the upside and G_d on the downside. The total diffuse outgoing radiative flux at P is called *radiosity* and denoted J_u on the upside and J_d on the downside. This radiosity is the sum of diffusively reflected radiation, emitted radiation and transmitted radiation coming from the other side of the semitransparent layer:

$$J_u = \rho_{d,u}G_u + \varepsilon_u e_{b,u}(T_u) \tag{4-97}$$

$$J_d = \rho_{d,d}G_d + \varepsilon_d e_{b,d}(T_d) \tag{4-98}$$

The net inward radiative heat fluxes on the upside and downside, q_u and q_d, are then given by the difference between the irradiation and the radiosity:

$$q_u = (1 - \rho_{s,u} - \tau_u)G_u - J_u \tag{4-99}$$

$$q_d = (1 - \rho_{s,d} - \tau_d)G_d - J_d \tag{4-100}$$

Bodies are considered to behave as ideal gray bodies, meaning that the absorptivity and emissivity are equal, and the reflectivity ρ_s is therefore obtained from the following relation:

$$\varepsilon_u + \rho_{d,u} = 1 - \rho_{s,u} - \tau_u \tag{4-101}$$

$$\varepsilon_d + \rho_{d,d} = 1 - \rho_{s,d} - \tau_d \tag{4-102}$$

Using Equation 4-97 to Equation 4-102, J_u and J_d can be eliminated and a general expression is obtained for the net inward heat fluxes into the semitransparent body based on G_u, G_d, T_u and T_d:

$$q_u = \varepsilon_u(G_u - e_{b,u}(T_u)) \tag{4-103}$$

$$q_d = \varepsilon_d(G_d - e_{b,d}(T_d)) \tag{4-104}$$

Thus, for ideal gray bodies, q is given by:

$$q = \varepsilon_u(G_u - e_{b,u}(T_u)) + \varepsilon_d(G_d - e_{b,d}(T_d)) \tag{4-105}$$

This is the expression used for the radiative boundary condition.

Incident rays which angle of incidence (measured between the ray and the normal to the surface) is higher than the critical angle are not transmitted, regardless the transmittance of the surface. They contribute to total reflection instead. Hence the directional transmissivity coefficient can be defined as

$$\tau(\theta) = \begin{cases} \tau \text{ if } \theta \le \theta_c \\ 0 \text{ if } \theta > \theta_c \end{cases}$$

where θ_c is the critical angle. Using the following relation

$$\rho_s(\theta) + \tau(\theta) = 1 - (\varepsilon + \rho_d)$$

we can establish

$$\rho_s(\theta) = \begin{cases} \rho_s & \text{if } \theta \le \theta_c \\ \rho_s + \tau & \text{if } \theta > \theta_c \end{cases}$$

Wavelength Dependence of Surface Properties

The surface properties for radiation, the emissivity, specular reflectivity and specular transmissivity can be dependent on the incident angle, the surface temperature, or the radiation wavelength. The emissivity, specular reflectivity and specular transmissivity are defined in Ref. 20.

The Surface-to-Surface Radiation interface in the Heat Transfer module implements the radiosity method that enables arbitrary temperature and wavelength dependent surface properties. The dependence of the surface properties on the incident angle can be considered by using the ray shooting method (Directional Dependence of Surface Properties).

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: $W/(m^3 \cdot sr)$), is denoted $e_{b, \lambda}(\lambda, T)$, and defined as (1-37 in Ref. 20):

$$e_{b, \lambda}(\lambda, T) = \frac{2\pi n^2 C_1}{\lambda^5 \left(e^{\frac{C_2}{\lambda T}} - 1 \right)} \tag{4-106}$$

where:

- the two constants C_1 (SI unit: $W \cdot m^2/sr$) and C_2 (SI unit: $m \cdot K$) 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 4-15 and Figure 4-16 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).
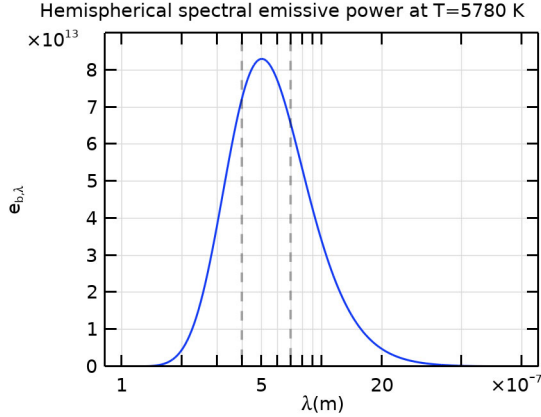
Figure 4-15: Planck distribution of a blackbody at 5780 K.
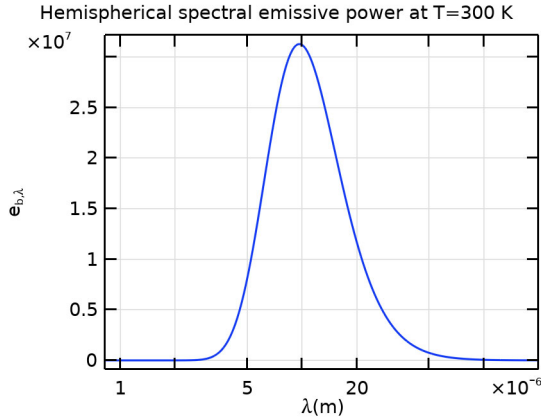
Figure 4-16: Planck distribution of a blackbody at 300 K.

The integral of $e_{b, \lambda}(\lambda, T)$ over a spectral band represents the power radiated on the spectral band and is defined by

$$\int_{\lambda_1}^{\lambda_2} e_{b, \lambda}(\lambda, T)d\lambda = F_{\lambda_1 T \rightarrow \lambda_2 T} \int_0^\infty e_{b, \lambda}(\lambda, T)d\lambda$$

where $F_{\lambda_1 T \rightarrow \lambda_2 T}$ is the fractional blackbody emissive power,

$$F_{\lambda_1 T \to \lambda_2 T} = \frac{\displaystyle\int_{\lambda_1}^{\lambda_2} e_{b, \lambda}(\lambda, T) d\lambda}{\displaystyle\int_{0}^{\infty} e_{b, \lambda}(\lambda, T) d\lambda}$$

Recall the Stefan-Boltzmann law that computes the power radiated across all wavelengths:

$$\int_{0}^{\infty} e_{b, \lambda}(\lambda, T) d\lambda = e_b(T) = n^2 \sigma T^4$$

where n is the refractive index, and σ is the Stefan-Boltzmann constant equal to $5.67 \cdot 10^{-8}$ W/(m$^2 \cdot$K^4). The power radiated in the spectral band $[\lambda_1, \lambda_2]$ becomes:

$$\int_{\lambda_1}^{\lambda_2} e_{b, \lambda}(\lambda, T) d\lambda = F_{\lambda_1 T \to \lambda_2 T} e_b(T)$$

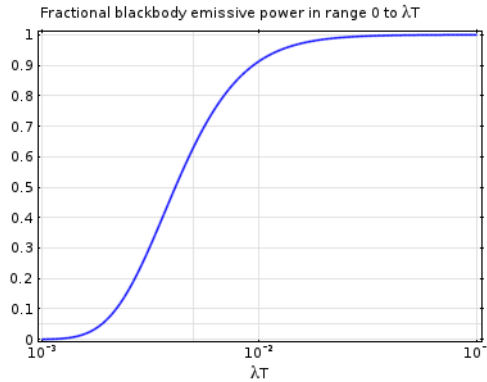|	
📋	The function $e_b(T)$ is available as a predefined function via `rad.feb(T)` in the Surface-to-Surface Radiation interface.

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.

Fractional blackbody emissive power in range 0 to λT

GRAY SURFACES

Gray surfaces correspond to the hypothesis that surface properties are independent of the radiation wavelength. This corresponds to Constant wavelength dependence in the Surface-to-Surface Radiation interface.

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.
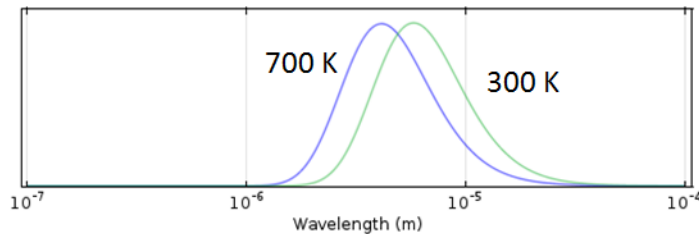
This setting is rarely applicable if there is solar radiation.

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. This corresponds to Solar and Ambient wavelength dependence in the Surface-to-Surface Radiation interface.

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 4-17).
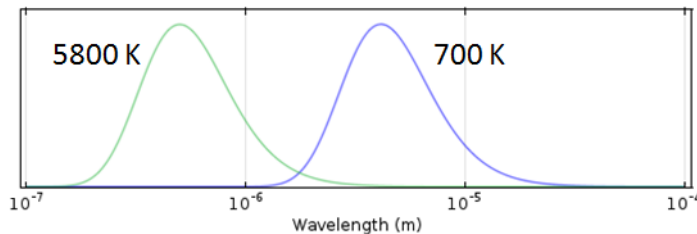
Figure 4-17: Normalized Planck distribution of blackbodies at 700 K and 5800 K.

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.
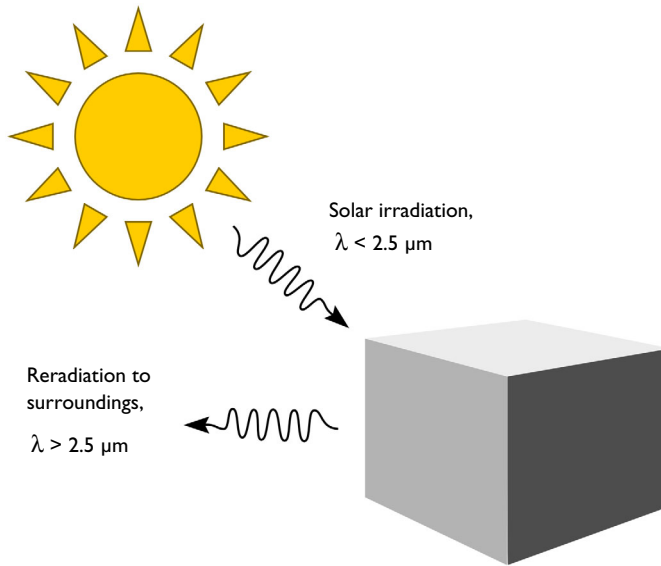
Figure 4-18: Absorption of solar radiation and emission to the surroundings.

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.
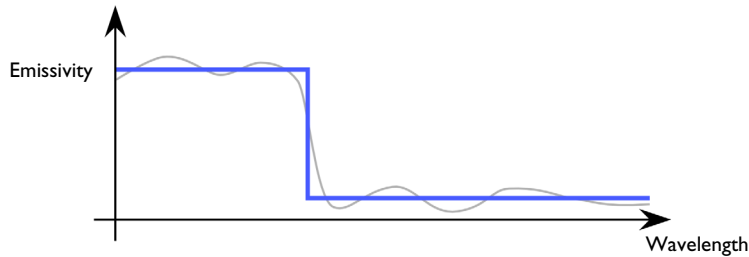
Figure 4-19: Solar and ambient spectral band approximation of the surface emissivity by a constant per band emissivity.

GENERAL SPECTRAL SURFACES

Spectral surfaces correspond to the hypothesis that surface properties are wavelength dependent. This corresponds to Multiple spectral bands wavelength dependence in the Surface-to-Surface Radiation interface.

The Heat Transfer Module supports constant surface properties per spectral bands and to adjust spectral intervals endpoints.
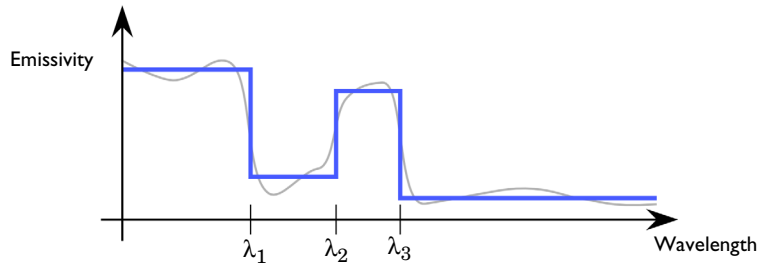
The multiple spectral bands approach is used in cases when the surface properties vary significantly over the bands of interest.

Directional Dependence of Surface Properties

The Surface-to-Surface Radiation interface in the Heat Transfer module implements the ray shooting method, that enables to set the dependence of the emissivity, of the specular reflectivity and of the specular transmissivity on the incident angle.

The polar and azimuthal angles, $\theta \in [0, \pi/2]$ and $\varphi \in [-\pi, \pi]$, are defined below:
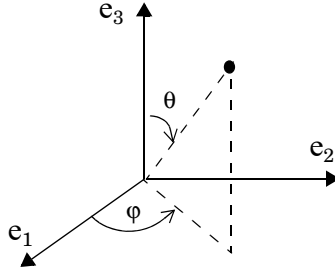
Figure 4-20: Definition of polar and azimuthal angles

The implementation supports directional surface properties that can be decomposed as the sum of two expressions. The first is defined as a function of θ and φ only, and the other as independent of θ and φ. The latter may depend on any other model quantity like the temperature, or the spatial coordinates.

- Emissivity

$$\varepsilon_{tot}(\theta, \varphi, T, \mathbf{x}) = f_\varepsilon(\theta, \varphi) + \varepsilon(T, \mathbf{x})$$

- Transmissivity

$$\tau_{tot}(\theta, \varphi, T, \mathbf{x}) = f_\tau(\theta, \varphi) + \tau(T, \mathbf{x})$$

- Specular reflectivity

$$\rho_{s, tot}(\theta, \varphi, T, \mathbf{x}) = 1 - \rho_d - \varepsilon_{tot}(\theta, \varphi, T, \mathbf{x}) - \tau_{tot}(\theta, \varphi, T, \mathbf{x})$$

Note that no dependence is defined for the diffuse reflectivity, ρ_d.

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 4-93, this source account for the difference between incident radiation, or *irradiation*, G, and radiation leaving the surface, or *radiosity*, J:

$$q = (1 - \rho_s)G - J$$

The radiosity, J, is given in Equation 4-92. It is the sum of diffusely reflected and emitted radiation. For diffuse-gray surfaces, J is defined by:

$$J = \rho_d G + \varepsilon e_b(T)$$

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:

$$G = G_m + G_{ext} + G_{amb} \tag{4-107}$$

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:

$$G_{amb} = \varepsilon_{amb} F_{amb} e_b(T_{amb})$$

- ε_{amb} is the *ambient emissivity;* (SI unit: 1), a dimensionless number in the range $0 \leq \varepsilon_{amb} \leq 1$. $\varepsilon_{amb} < 1$ means that part of the energy coming from radiative bodies is not absorbed by the ambient air.

- 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, $0 \leq F_{amb} \leq 1$ 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 (Surface-to-Surface Radiation Interface) 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:

$$J = \rho_d(G_m(J) + G_{ext} + G_{amb}) + \varepsilon e_b(T) \qquad (4\text{-}108)$$

- Diffuse Mirror (Surface-to-Surface Radiation Interface) 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. 21). It resembles a mirror that absorbs all irradiation and then radiates it back in all directions.

- Prescribed Radiosity (Surface-to-Surface Radiation Interface) 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.

- Opaque Surface (Surface-to-Surface Radiation Interface) is available when the surface-to-surface radiation method is Ray shooting. It accounts for specular reflection. The conservation equation reads

$$\varepsilon + \rho_d + \rho_s = 1,$$

and the radiosity reads as in Equation 4-108.

- Semitransparent Surface (Surface-to-Surface Radiation Interface) is available when the surface-to-surface radiation method is Ray shooting. It accounts for reflection, transmission and the conservation equation reads

$$\varepsilon + \rho_d + \rho_s + \tau = 1,$$

and the radiosities read

$$J_u = (\rho_{d,u}(G_{m,u}(J_u) + G_{ext,u} + G_{amb,u}) + \varepsilon_u e_b(T))$$
$$+ \tau_d(G_{m,d}(J_d) + G_{ext,d} + G_{amb,d}) \tag{4-109}$$

$$J_d = (\rho_{d,d}(G_{m,d}(J_d) + G_{ext,d} + G_{amb,d}) + \varepsilon_d e_b(T))$$
$$+ \tau_d(G_{m,u}(J_u) + G_{ext,u} + G_{amb,u}) \tag{4-110}$$

The Surface-to-Surface Radiation interface handles the radiosity J as a shape function unless J is prescribed.

> 💡 Whereas diffuse and specular reflectivities are handled by the different features of the Surface-to-Surface Radiation interface, only diffuse emissivity is considered.

The Radiosity Method for Diffuse-Spectral Surfaces

For a general diffuse-spectral surface:

$$J = \int_0^\infty (\rho_d(\lambda, T)G(\lambda) + \varepsilon(\lambda, T)e_{b,\lambda}(\lambda, T))d\lambda$$

where

- $\varepsilon(\lambda, T)$ and $\rho_d(\lambda, T)$ are is the hemispherical spectral surface emissivity and diffuse reflectivity, dimensionless quantities in the range [0,1]. Diffuse-spectral surface corresponds to a surface properties are 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·sr)) defined in Equation 4-106.

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:

$$
J = \sum_{i=1}^{N} J_i
$$

$$
J_i = \rho_{d,i} G_i + \varepsilon_i e_b(T)
$$

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

- Surface diffuse reflectivity on B_i: $\rho_{d,i}(T) = \rho_d(\lambda, T)$ 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}:

$$
G_{amb,i} = \int_{\lambda = \lambda_{i-1}}^{\lambda_i} G_{amb}(\lambda) d\lambda = F_{\lambda_{i-1}T \to \lambda_i T} F_{amb,i} \varepsilon_{amb,i} e_b(T_{amb})
$$

- External radiation sources on B_i with $q_{0,s,i}$ and $P_{s,i}$ the external radiation source heat flux and heat rate, respectively, over B_i:

$$
G_{ext,i} = \int_{\lambda = \lambda_{i-1}}^{\lambda_i} G_{ext}(\lambda) d\lambda = F_{ext,i}(i_s) q_{0,s,i}
$$

or

$$
G_{ext,i} = \int_{\lambda = \lambda_{i-1}}^{\lambda_i} G_{ext}(\lambda) d\lambda = F_{ext,i}(i_s) P_{s,i}
$$

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, $q_{0,s}$, and heat rate, P_s, over all wavelengths:

$$
G_{ext,i} = F_{ext,i} F_{\lambda_{i-1}T \to \lambda_i T}(i_s) q_{0,s}
$$

or

$$
G_{ext,i} = F_{ext,i} F_{\lambda_{i-1}T \to \lambda_i T}(i_s) P_s
$$

The Surface-to-Surface Radiation Interface includes the following radiation types:

- Diffuse Surface (Surface-to-Surface Radiation Interface) 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: W/m^2), at every other point in view. The radiosity, in turn, is a function of $G_{m, i}$, which leads to an implicit radiation balance:

$$J_i = \rho_{d, i}(G_{m, i}(J_i) + G_{ext, i} + G_{amb, i}) + \varepsilon_i e_b(T) \qquad (4\text{-}111)$$

- Diffuse Mirror (Surface-to-Surface Radiation Interface) 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. 21). It resembles a mirror that absorbs all irradiation and then radiates it back in all directions.

- Prescribed Radiosity (Surface-to-Surface Radiation Interface) makes it possible to specify the surface radiation for each spectral band. Using the graybody radiation definition, the radiosity is then $F_{\lambda_1 T \to \lambda_2 T} e_b(T)$. A user-defined surface radiosity expression can also be defined.

- Opaque Surface (Surface-to-Surface Radiation Interface) is available with Ray shooting as Surface-to-surface radiation method. It accounts for specular reflection. The conservation equation reads

$$\varepsilon_i + \rho_{d, i} + \rho_{s, i} = 1$$

and the radiosity reads as in Equation 4-111.

- Semitransparent Surface (Surface-to-Surface Radiation Interface) is available with Ray shooting as Surface-to-surface radiation method. It accounts for reflection, transmission and the conservation equation reads

$$\varepsilon_i + \rho_{d, i} + \rho_{s, i} + \tau_i = 1$$

and the radiosities read

$$
\begin{aligned}
J_{u, i} = {} & \rho_{d, u, i}(G_{mu, i}(J_{u, i}) + G_{ext, u, i} + G_{amb, u, i}) + \varepsilon_{u, i} e_b(T) \\
& + \tau_{d, i}(G_{md, i}(J_{d, i}) + G_{ext, d, i} + G_{amb, d, i})
\end{aligned} \qquad (4\text{-}112)
$$

$$
\begin{aligned}
J_{d, i} = {} & \rho_{d, u, i}(G_{md, i}(J_{d, i}) + G_{ext, d, i} + G_{amb, d, i}) + \varepsilon_{d, i} e_b(T) \\
& + \tau_{u, i}(G_{mu, i}(J_{u, i}) + G_{ext, u, i} + G_{amb, u, i})
\end{aligned} \qquad (4\text{-}113)
$$

The Surface-to-Surface Radiation interface handles the radiosity J_i as a shape function unless J_i is prescribed.

> Whereas diffuse and specular reflectivities are handled by the different features of the Surface-to-Surface Radiation interface, only diffuse emissivity is considered.

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 4-108 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

$$J_{amb} = e_b(T_{amb})$$

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 4-21. 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}.
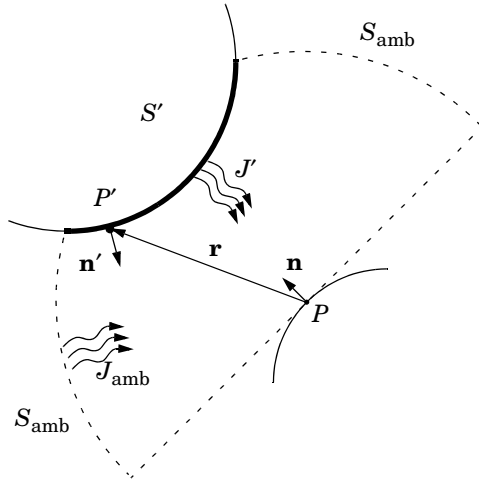

Figure 4-21: Example geometry for surface-to-surface radiation.

The mutual irradiation at point P is given by the following surface integral:

$$G_m = \int_{S'} \frac{(-\mathbf{n}' \cdot \mathbf{r})(\mathbf{n} \cdot \mathbf{r})}{\pi |\mathbf{r}|^4} J' ds$$

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':

$$F_{amb} = 1 - F' = 1 - \int_{S'} \frac{(-\mathbf{n}' \cdot \mathbf{r})(\mathbf{n} \cdot \mathbf{r})}{\pi |\mathbf{r}|^4} ds$$

The two last equations plug into Equation 4-107 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

$$G_{\mathrm{m}} = \int_{S_\perp'} \frac{(-\mathbf{n}' \cdot \mathbf{r}_\perp)(\mathbf{n} \cdot \mathbf{r}_\perp)}{2|\mathbf{r}_\perp|^3} J' ds \qquad (4\text{-}114)$$

$$F_{\mathrm{amb}} = 1 - \int_{S_\perp'} \frac{(-\mathbf{n}' \cdot \mathbf{r}_\perp)(\mathbf{n} \cdot \mathbf{r}_\perp)}{2|\mathbf{r}_\perp|^3} ds$$

where the integral over $S_\perp'$ denotes the line integral along the boundaries of the 2D geometry.

In axisymmetric geometries or when a symmetry plane 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 4-114. For cases with specularly reflective surfaces, specular view factors depends also on specular reflectivities of surrounding surfaces, see Ref. 23.

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 the following surface-to-surface radiation methods, which are selected in the Radiation Settings section in a Heat Transfer interface:

• Hemicube
• Direct area integration
• Ray shooting

> 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. 20. 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:

$$i_{sx} = -\cos(\varphi_s)\sin(\theta_s)$$
$$i_{sy} = \sin(\varphi_s)\sin(\theta_s)$$
$$i_{sz} = -\cos(\theta_s)$$

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

In this section:

- Radiation and Participating Media Interactions
- Radiative Transfer Equation
- Boundary Condition for the Radiative Transfer Equation
- Heat Transfer Equation in Participating Media
- Discrete Ordinates Method (DOM)
- Discrete Ordinates Method Implementation in 2D
- P1 Approximation Theory
- Radiation in Absorbing-Scattering Media Theory
- Polychromatic Radiation
- Radiative Beam in Absorbing Media Theory
- Rosseland Approximation Theory

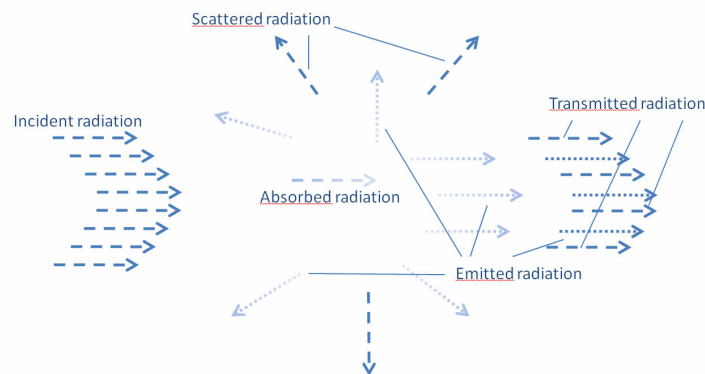## *Radiation and Participating Media Interactions*

Figure 4-22: 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. 23):

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi}\int_{4\pi} I(\Omega')\phi(\Omega', \Omega)d\Omega' \qquad (4\text{-}115)$$

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

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi} \qquad (4\text{-}116)$$

📝	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)

SCATTERING PHASE FUNCTION

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 12th order:

$$\phi(\mu_0) = 1 + \sum_{n=1}^{12} a_n P_n(\mu_0)$$

where P_n are the nth order Legendre polynomials.

Legendre polynomials can be defined by the Rodriguez formula:

$$P_k(x) = \frac{1}{2^k k!} \frac{d^k}{dx^k}((x^2 - 1)^k)$$

- *Henyey–Greenstein* phase function:

$$\phi(\mu_0) = \frac{1}{K} \cdot \frac{1 - \eta^2}{(1 + \eta^2 - 2\eta\mu_0)^{3/2}}$$

where $-1 < \eta < 1$ is the anisotropy parameter and K is defined as follows to produce a normalized phase function:

$$K = \frac{1}{4\pi} \cdot \int_{4\pi} \frac{1 - \eta^2}{(1 + \eta^2 - 2\eta\mu_0)^{3/2}} d\Omega$$

INCIDENT RADIATION

A quantity of interest is the incident radiation, denoted G (SI unit: W/m^2), 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

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi} \tag{4-117}$$

- Equation 4-117 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 coming from the domain and 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 $I(\Omega) = I_b(T)$.

The net heat flux corresponding to the balance between the energy received by the surface and emitted by the surface is defined by

$$q_{r,\,net} = q_{r,\,in} - q_{r,\,out}$$

This net heat flux accounts for the radiation coming from the domain where the RTE is solved for, the radiation coming from the exterior is not modeled and hence not accounted for.

If the surface is semitransparent, the radiative intensity transmitted diffusively or specularly. For an external boundary the external intensity, I_{ext}, is transmitted in the domain. The radiative intensity entering participating media along the Ω direction is

$$I(\Omega) = \varepsilon I_b(T) + \frac{\rho_d}{\pi}q_{r,\,out} + \frac{\tau_d}{\pi}q_{r,\,in,\,ext} + \tau_s I_{ext}$$

for all Ω such that $\mathbf{n} \cdot \Omega < 0$,

where

- ρ_d is the surface diffuse reflectivity,
- τ_d is the surface diffuse transmissivity,
- τ_s is the surface specular transmissivity,
- I_{ext} is the external radiation intensity,
- $q_{r,in}$ is the heat flux striking the wall from exterior:

$$q_{r,\,in,\,ext} = \int_{\mathbf{n} \cdot \Omega \le 0} I_{ext}(\Omega)(\mathbf{n} \cdot \Omega)d\Omega$$

The emissivity, diffuse reflectivity, diffuse transmissivity, and specular transmissivity satisfy the following relation:

$$\varepsilon + \rho_d + \tau_d + \tau_s = 1$$

The net heat flux corresponding to the balance between the energy received by the surface and emitted by the surface is defined by

$$q_{r, \, net} = q_{r, \, in} - q_{r, \, out} + \varepsilon_{ext} q_{r, \, in, \, net} - \varepsilon_{ext} \pi \varepsilon I_b(T)$$

For semitransparent surfaces, the net heat flux accounts for the radiation coming from the domain and from the exterior.

The definition of the radiative intensity on internal boundaries is similar to the definition for external boundaries except that the external radiation is replaced by the radiation exiting the adjacent domain.

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. 23):

$$Q_r = \nabla \cdot \mathbf{q_r} = \kappa(G - 4\pi I_b(T))$$

In order to couple radiation in participating media, radiative heat flux is taken into account in addition to conductive heat flux. Recalling Equation 4-18, 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:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot \mathbf{q} = \kappa(G - 4n\sigma T^4) + \alpha_p T \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right) + \tau : \nabla \mathbf{u} + Q$$

Discrete Ordinates Method (DOM)

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 provides a discretization of angular space. The quadrature sets Level Symmetric Even, Level Symmetric Hybrid and Equal Weight Odd are designed using moment-matching conditions (see Ref. 24) and yield S_N approximations with $N(N+2)$ directions in 3D or $N(N+2)/2$ in 2D (see Ref. 23). The quasi-uniform quadrature set yields T_N approximations with $8N^2$ directions in 3D or $4N^2$ in 2D (Ref. 46, Ref. 47). These approximations are not designed using moment-matching conditions but allow for higher-order discretization compared with S_N approximations.

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 $I_1, I_2, \ldots, I_n$.

RADIATION IN PARTICIPATING MEDIA

Each dependent variable satisfies the equation

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^{n} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i)$$

subject to appropriate boundary conditions, where $\mathbf{S}_i$ is the ith discrete direction.

OPAQUE SURFACE

The node Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) defines the default boundary condition for radiative intensities I_1, I_2, ..., I_n:

$$I_{i,\,\text{bnd}} = \varepsilon I_\text{b}(T) + \frac{1-\varepsilon}{\pi} q_{\text{r, out}} \qquad \text{for all } \mathbf{S}_i \text{ such that } \mathbf{n} \cdot \mathbf{S}_i < 0 \qquad (4\text{-}118)$$

with

$$q_{\text{r, out}} = \sum_{\mathbf{n} \cdot \mathbf{S}_j > 0} w_j I_j (\mathbf{n} \cdot \mathbf{S}_j)$$

In Equation 4-118, the first term in the right hand side is the emitted radiative intensity, while the second term represents the reflected radiative intensity.

SEMITRANSPARENT SURFACE

The node Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) defines the following boundary condition for radiative intensities I_1, I_2, ..., I_n:

$$I_{i,\,\text{bnd}} = \varepsilon I_\text{b}(T) + \frac{\rho_\text{d}}{\pi} q_{\text{r, out}} + \frac{\tau_\text{d}}{\pi} q_{\text{r, out, ext}} + \tau_\text{s} I_{i,\,\text{ext}} \qquad (4\text{-}119)$$

for all $\mathbf{S}_i$ such that $\mathbf{n} \cdot \mathbf{S}_i < 0$, with

$$q_{\text{r, out}} = \sum_{\mathbf{n} \cdot \mathbf{S}_j > 0} w_j I_j (\mathbf{n} \cdot \mathbf{S}_j)$$

and

$$q_{\text{r, out, ext}} = - \sum_{\mathbf{n} \cdot \mathbf{S}_j \le 0} w_j I_{j,\,\text{ext}} (\mathbf{n} \cdot \mathbf{S}_j)$$

In Equation 4-119, the right hand side is composed of the emitted radiative intensity, the reflected radiative intensity, and the radiative intensity transmitted in a diffuse and specular way.

INCIDENT INTENSITY

The Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) node defines a boundary that receives incident radiative intensity I_{ext} and that is transparent for outgoing intensity:

$$I_{i, \text{bnd}} = I_{\text{ext}} \qquad \text{for all } \mathbf{S}_i \text{ such that } \mathbf{n} \cdot \mathbf{S}_i < 0$$

RADIATIVE SOURCE

The node Radiative Source accounts for a directional power density I_i in the radiative transfer equation:

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^{n} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) + q_i \cdot$$

Discrete Ordinates Method Implementation in 2D

For a given index i, define two indices, i^+ and i^-, so that

- $\Omega, \mathbf{S}_{i+}$, 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:

$$\mathbf{S}_{i^+} \cdot \nabla I_{i^+} = \kappa I_b(T) - \beta I_{i^+} + \frac{\sigma_s}{4\pi} \sum_{j=1}^{n} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i^+})$$

$$\mathbf{S}_{i^-} \cdot \nabla I_{i^-} = \kappa I_b(T) - \beta I_{i^-} + \frac{\sigma_s}{4\pi} \sum_{j=1}^{n} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i^-})$$

Halving the sum of the two equations above and using $I_{i^+} = I_{i^-} = \tilde{I}_i$ ($I_{i^-} = I_{i^+}$ in 2D) yields

$$\tilde{\mathbf{S}}_i \cdot \nabla \tilde{I}_i = \kappa I_b(T) - \beta \tilde{I}_i + \frac{\sigma_s}{8\pi} \sum_{j=1}^{n} w_j I_j (\phi(\mathbf{S}_j, \mathbf{S}_{i^+}) + \phi(\mathbf{S}_j, \mathbf{S}_{i^-}))$$

which may recast

$$\tilde{\mathbf{S}}_i \cdot \nabla \tilde{I}_i = \kappa I_{\mathrm{b}}(T) - \beta \tilde{I}_i \quad + \frac{\sigma_{\mathrm{s}}}{8\pi} \sum_{j=1}^{n/2} w_{j^+} I_{j^+} (\phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^+}) + \phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^-}))$$

$$+ \frac{\sigma_{\mathrm{s}}}{8\pi} \sum_{j=1}^{n/2} w_{j^-} I_{j^-} (\phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^+}) + \phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^-}))$$

or

$$\tilde{\mathbf{S}}_i \cdot \nabla \tilde{I}_i = \kappa I_{\mathrm{b}}(T) - \beta \tilde{I}_i \quad + \frac{\sigma_{\mathrm{s}}}{8\pi} \sum_{j=1}^{n/2} w_{j^+} I_{j^+} \phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^+}) + w_{j^-} I_{j^-} \phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^-})$$

$$+ \frac{\sigma_{\mathrm{s}}}{8\pi} \sum_{j=1}^{n/2} w_{j^+} I_{j^+} \phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^-}) + w_{j^-} I_{j^-} \phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^+})$$

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 the 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:

$$\tilde{\mathbf{S}}_i \cdot \nabla \tilde{I}_i = \kappa I_{\mathrm{b}}(T) - \beta \tilde{I}_i + \frac{\sigma_{\mathrm{s}}}{4\pi} \sum_{j=1}^{n/2} \tilde{w}_j \tilde{I}_j \phi(\tilde{\mathbf{S}}_j, \tilde{\mathbf{S}}_i) \qquad (4\text{-}120)$$

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

$$\int_{4\pi} I(\Omega)d\Omega \approx \sum_{j=1}^{n} w_j I_j = \sum_{j=1}^{n/2} w_j I_j + w_{j^+} I_{j^+} = \sum_{j=1}^{n/2} \tilde{w}_i \tilde{I}_i \qquad (4\text{-}121)$$

with $\tilde{w}_i = 2w_i$.

Using results from Equation 4-120 and Equation 4-121 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.

P1 Approximation Theory

The **P1 approximation** is available as a radiation discretization method in The Radiation in Participating Media Interface.

The 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: $\tau \gg 1$, 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 the incident radiation G (SI unit: W/m^2), 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

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_{4\pi} I(\Omega')\phi(\Omega', \Omega)d\Omega'$$

by solving following equation for $G = \int_{4\pi} I(\Omega)d\Omega$ (Ref. 23):

$$\nabla \cdot (-D_{\mathrm{P1}}\nabla G) = -Q_{\mathrm{r}} \qquad (4\text{-}122)$$

where

• D_{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:

$$Q_{\mathrm{r}} = \kappa(G - 4\pi I_{\mathrm{b}}) \qquad (4\text{-}123)$$

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. 23):

$$\mathbf{n} \cdot (-D_{\mathrm{P1}}\nabla G) = -q_{\mathrm{r, net}}$$

where $\mathbf{q}_{\mathrm{r, net}}$ is the net radiative heat flux at the boundary.

RADIATION IN PARTICIPATING MEDIA

For the Participating Medium (Radiation in Participating Media Interface) feature node, the equation Equation 4-122 is implemented.

In addition Q_{r}, defined by Equation 4-123, 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 (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) boundary condition defines a boundary opaque to radiation and defines the incident intensity on a boundary:

$$\mathbf{n} \cdot (-D_{P1}\nabla G) = q_{r,\,net}$$

The Opaque Surface feature accounts for the net radiative heat flux, $\mathbf{q}_{r,\,net}$, 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: $\varepsilon = 1$

Gray Wall
The radiative heat flux at the boundary depends on the surface emissivity, ε:

$$q_{r,\,net} = \frac{\varepsilon}{2(2-\varepsilon)}(4\pi I_{b,\,w} - G)$$

with

$$I_{b,\,w} = I_b = \frac{n^2 \sigma T^4}{\pi}$$

Black Wall
The radiative heat flux at the boundary expression simplifies to

$$\mathbf{n} \cdot (-D_{P1}\nabla G) = -\frac{1}{2}(4\pi I_{b,\,w} - G),$$

with

$$I_{b,\,w} = I_b = \frac{n^2 \sigma T^4}{\pi}$$

SEMITRANSPARENT SURFACE

The Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) boundary condition defines a boundary that emits radiation, and reflects one part of the incident intensity while the remaining is transmitted diffusively.

The net radiative heat flux, $\mathbf{q}_{r,\,net}$, accounted for in the heat balance, is defined as

$$q_{r,\,net} = \frac{2}{1+\rho_d}(\pi\varepsilon I_{b,\,w} + \tau_d G) - \frac{1-\rho_d}{2(1+\rho_d)}G$$

where ρ_d is the surface diffuse reflectivity, and τ_d is the surface diffuse transmissivity.

The Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) 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 (incident radiation), $\mathbf{q}_{r,\,net}$ (net radiative heat flux) and I_{ext} (incident radiative intensity) is

$$G + 2q_{r,\,net} = 4\int_{\Omega \cdot \mathbf{n} < 0} I_{ext}(\Omega)\Omega \cdot \mathbf{n}d\Omega$$

by defining

$$I_{ext} = \int_{\Omega \cdot \mathbf{n} < 0} I_{ext}(\Omega)\Omega \cdot \mathbf{n}d\Omega$$

there is

$$q_{r,\,net} = \frac{1}{2}(4\pi I_{ext} - G)$$

which defines the heat radiative heat flux and also contributes to G boundary condition:

$$\mathbf{n} \cdot (-D_{P1}\nabla G) = -q_{r,\,net}$$

RADIATIVE SOURCE

The node Radiative Source accounts for a power density Q in the P1 approximation:

$$\nabla \cdot (-D_{P1}\nabla G) = -Q_r + Q \, .$$

Radiation in Absorbing-Scattering Media Theory

The Radiation in Absorbing-Scattering Media Interface is available in 2D, 2D axisymmetric, and 3D components to model the propagation, attenuation, and scattering of an incident light within a semitransparent material considered to be non-emitting.

The radiative intensity $I(\Omega)$ at a given position following the Ω direction is the solution of the radiative transfer equation with no emission term (see Ref. 23):

$$\Omega \cdot \nabla I(\Omega) = -\beta I(\Omega) + \frac{\sigma_s}{4\pi}\int_{4\pi} I(\Omega')\phi(\Omega', \Omega)d\Omega' \qquad \text{(4-124)}$$

where

- 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)

See Radiative Transfer Equation for details about the phase function, $\phi(\Omega', \Omega)$.

The incident radiation, denoted G (SI unit: W/m^2), is defined by

$$G = \int_{4\pi} I(\Omega)d\Omega$$

If the Discrete Ordinates Method (DOM) is used for the approximation of Equation 4-124, G is computed as

$$G = \sum_{i=1}^{N} \omega_i I_i$$

and

$$\mathbf{S}_i \cdot \nabla I_i = -\beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^{N} \omega_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i)$$

where

- $\mathbf{S}_i$ is the ith discrete ordinate.
- I_i is the ith component of the radiative intensity.
- ω_j is the ith quadrature weight.

If the P1 Approximation Theory is used instead, G is the solution of the following equation,

$$\nabla \cdot (-D_{P1} \nabla G) = -\kappa G, \qquad\qquad (4\text{-}125)$$

with the following boundary condition for an opaque surface,

$$\mathbf{n} \cdot (-D_{P1} \nabla G) = \frac{\varepsilon}{2(2-\varepsilon)} G,$$

where D_{P1} is the P1 diffusion coefficient and ε the surface emissivity.

Polychromatic Radiation

As in the Surface-to-Surface Radiation Interface, polychromatic radiation is modeled using the stepwise-gray model (Ref. 23) to account for wavelength dependence of emission, absorption and scattering through material properties, see the section Wavelength Dependence of Surface Properties.

In the Discrete Ordinates Method (DOM), the Radiative Transfer Equation is solved for each spectral band:

$$\mathbf{S}_i \cdot \nabla I_{i,k} = \kappa_k I_b(T)\text{FEP}_k(T) - \beta_k I_{i,k} + \frac{\sigma_{s,k}}{4\pi} \sum_{j=1}^{N} \omega_j I_{j,k} \phi_k(\mathbf{S}_j, \mathbf{S}_i),$$

where FEP is the fractional emissive power, also presented in the section Wavelength Dependence of Surface Properties.

If the P1 Approximation Theory is used instead, the incident radiation G_k for the spectral band k is solution of the following equation,

$$\nabla \cdot (-D_{\text{P1},k} \nabla G_k) = -Q_{r,k}, \tag{4-126}$$

where

- $D_{\text{P1},k}$ is the P1 diffusion coefficient, defined as

$$D_{\text{P1},k} = \frac{1}{3\kappa_k + \sigma_{s,k}(3 - a_{1,k})}$$

- $a_{1,k}$ is the linear Legendre coefficient of the scattering phase function
- $Q_{r,k}$ is the radiative heat source:

$$Q_{r,k} = \kappa_k(G_k - 4\pi I_b\text{FEP}_k(T)). \tag{4-127}$$

Radiative Beam in Absorbing Media Theory

The Radiative Beam in Absorbing Media Interface is available in 2D, 2D axisymmetric, and 3D components to model the propagation and the attenuation of an incident light within a semitransparent material. The radiative intensity of the beam is described by

the Beer-Lambert Law, which corresponds to the radiative transport equation (RTE) under the following hypothesis:

- The radiative beam in the absorbing medium is collimated and each beam propagates always in the same direction.
- The light experiences no refraction, reflection, or scattering within the material itself.
- There is no significant emission of the material in the wavelength range of the incident light. This applies well to laser beams, whose wavelength is in general much shorter than the one of the radiation emitted by the medium.

In these conditions, the radiative intensity I_i (SI unit: W/m^2) of the ith beam through the material decreases as the beam propagates and is absorbed by the medium. This is described by the Beer-Lambert Law equation:

$$\frac{\mathbf{e}_i}{\|\mathbf{e}_i\|} \cdot \nabla I_i = -\kappa I_i$$

where $\mathbf{e}_i$ is the orientation of the ith beam, and κ is the absorption coefficient (SI unit: m^{-1}) of the medium.

The radiative heat source Q_r (SI unit: W/m^3), corresponding to the energy deposited by the radiative beam, is defined by:

$$Q_r = \sum_i \kappa I_i.$$

Rosseland Approximation Theory

The Rosseland approximation is available as a radiation discretization method as part of the Optically Thick Participating Medium subnode.

Rosseland approximation relies on the hypotheses that the participating medium is optically thick — that is, $\tau \gg 1$ — 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 cost 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. 23):

$$q_{r,\lambda} = -\frac{4\pi}{\beta_\lambda}\nabla i_{b,\lambda}$$

For a gray media it leads to

$$q_r = -\frac{4\sigma}{3\beta_R}\nabla(n^2 T^4)$$

Assuming a constant refractive index, this can be rewritten as $q_r = -k_R \nabla T$ with

$$k_R = \frac{16n^2\sigma T^3}{3\beta_R}$$

and

$$q_r = -\frac{16n^2\sigma T^3}{3\beta_R}\nabla T$$

Rosseland mean extinction coefficient, β_R, can be defined from the absorption and scattering coefficients of the media from $\beta_R = \kappa + \sigma_s$.

Theory for Moisture Transport

Theory for Moisture Transport in Air

In moist air, the transport of moisture is done by convection and binary diffusion of vapor in air.

Two formulations are available, depending on the magnitude of moist air density variations with moisture content.

In both formulations, the moisture is supposed to be constituted only of vapor when considering moist air. In other terms the liquid concentration is null, except on boundaries where condensation may take place.

DILUTED SPECIES FORMULATION

With this formulation, it is supposed that the spatial and temporal variations of the vapor concentration are small enough not to induce significant variations of the moist air density.

The Moisture Transport in Air Interface solves for the following equation, in which the moisture content variation is expressed through the transport of vapor concentration, c_v, by convection and binary diffusion in air:

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w = G \qquad (4\text{-}128)$$

$$\mathbf{g}_w = -M_v D \nabla c_v$$

$$c_v = \phi_w c_{sat}$$

with the following material properties, fields, and source:

- M_v (SI unit: kg/mol) is the molar mass of water vapor.
- ϕ_w (dimensionless) is the relative humidity.
- c_{sat} (SI unit: mol/m^3) is the vapor saturation concentration.
- D (SI unit: m^2/s) is the vapor diffusion coefficient in air.
- $\mathbf{u}$ (SI unit: m/s) is the air velocity field.
- $\mathbf{g}_w$ (SI unit: kg/(m^2·s)) is the vapor diffusive flux.
- G (SI unit: kg/(m^3·s)) is the moisture source (or sink).

When the spatial and temporal variations of the vapor concentration induce significant variations of the moist air density, the moisture content variation is expressed through the transport of vapor mass fraction, ω_v, defined as:

$$\omega_v = \frac{M_v c_v}{\rho_g}$$

where ρ_g is the moist air density (SI unit: kg/m^3).

The Heat Transfer in Moist Air Interface solves for the following equation:

$$\rho_g \frac{\partial \omega_v}{\partial t} + \rho_g \mathbf{u} \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w = G \qquad (4\text{-}129)$$

$$\mathbf{g}_w = -\rho_g D \nabla \omega_v$$

This is equivalent to solve the following equations on the vapor concentration:

$$M_v \rho_g \frac{\partial(c_v / \rho_g)}{\partial t} + M_v \rho_g \mathbf{u} \cdot \nabla(c_v / \rho_g) + \nabla \cdot \mathbf{g}_w = G$$

$$\mathbf{g}_w = -M_v \rho_g D \nabla(c_v / \rho_g)$$

By neglecting the spatial and temporal variations of ρ_g in the equations above, the equation of the diluted species formulation, Equation 4-128, is obtained.

Theory for Moisture Transport in Porous Media

In a porous material, moisture may be present under vapor and liquid state, and moisture transport occurs through convection and diffusion, liquid water being fixed on the pores walls mainly in two ways:

- Adsorption: the vapor is fixed under molecular films on the pores walls. Equilibrium vapor pressure is reached in the pores under thermodynamic conditions that are significantly different than those necessary in a free medium. Thus saturation conditions are changed, and the relative humidity is significantly changed as well, becoming a relevant variable for the characterization of moisture content.

- Capillarity: when the porous matrix is wetted by liquid water, there is a pressure difference between the liquid and gas (moist air) phases, due to the curvature of the wetting interfaces in each pore. The pressure difference, called capillary pressure,

becomes the driving potential for a further filling of the pores by the water. Note that in some applications, the capillary pressure is replaced by the suction.

The predominance of adsorption or capillarity depends both on the amount of liquid water present in the porous medium, and on the size and shape of the pores, and should be defined in function of the material and process under study. Two moisture regions can be distinguished:

- Hygroscopic region: adsorption is the predominant phenomenon, and the equilibrium moisture content is better characterized from the relative humidity, through the specification of a sorption curve. For relative humidity values below 0.95, the slope of the sorption curve is relatively small, and the moisture content increases in a nearly linear way with relative humidity. Building materials are most of the time in this moisture region. Above 0.95, a very small variation of relative humidity induces a large variation of the moisture content, and the capillary pressure is best suited to characterize the moisture content.

- Capillary water region: capillarity is the predominant phenomenon, and the equilibrium moisture content is better characterized from the capillary pressure, through the specification of a retention curve (the van Genuchten and Brooks & Corey retention models for example).

The relative humidity and capillary pressure are called state variables: they describe the thermodynamic state of water in the pore space under two phases in equilibrium. They are related through Kelvin's law, which is a relation of thermodynamical origin. As long as equilibrium is satisfied, the state variables are continuous between two media (at the interface between two porous media with different porosities, or between a porous and a free medium), and Kelvin's law is valid on both sides of the interface. In opposition, the liquid saturation or the moisture content depend on the porous matrix.

Note that capillary transport of liquid water should be considered in both regions.

The Moisture Transport in Porous Media Interface is dedicated to the modeling of porous media in the hygroscopic moisture region. The total moisture content is thus defined by a sorption curve as

$$\varepsilon_p s_l \rho_l + \varepsilon_p \rho_g \omega_v (1 - s_l) \ = \ w(\phi_w)$$

where the liquid saturation, s_l, describes the amount of liquid water within the pores, and ε_p (dimensionless) is the porosity.

By summing the mass conservation equations for liquid water and vapor, a single equation for the transport of the total moisture can be written, in which the evaporation and condensation source terms of each equation cancel out:

$$\frac{\partial w(\phi_w)}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w + \mathbf{u}_l \cdot \nabla \rho_l + \nabla \cdot \mathbf{g}_{lc} = G \qquad (4\text{-}130)$$

with the following material properties, fields, and source:

- $w(\phi_w)$ (SI unit: kg/m^3) is the total moisture content.
- ϕ_w (dimensionless) is the relative humidity.
- ρ_g (SI unit: kg/m^3) is the moist air density, defined from the dry air and vapor densities, in function of the amount of vapor.
- $\mathbf{u}_g$ (SI unit: m/s) is the moist air velocity field, that should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area.
- ω_v (dimensionless) is the vapor mass fraction in moist air, defined as

$$\omega_v = \frac{M_v \phi_w c_{sat}(T)}{\rho_g}$$

- M_v (SI unit: kg/mol) is the molar mass of water vapor.
- c_{sat} (SI unit: mol/m^3) is the vapor saturation concentration.
- T (SI unit: K) is the temperature.
- $\mathbf{g}_w$ is the moisture diffusive flux is defined as:

$$\mathbf{g}_w = -\rho_g D_{eff} \nabla \omega_v$$

- D_{eff} (SI unit: m^2/s) is the effective vapor diffusion coefficient in the porous medium, computed from the diffusion coefficient in a free medium, and accounting for the porosity and tortuosity of the porous medium.
- $\mathbf{u}_l$ (SI unit: m/s) is the liquid water velocity field, defined from the absolute pressure gradient by the Darcy's law as:

$$\mathbf{u}_l = -\frac{\kappa_{rl}\kappa}{\mu_l}\nabla p_A$$

- κ_{rl} (dimensionless) is the relative liquid water permeability, that may be defined as a function of the liquid saturation.
- κ (SI unit: m^2) is the porous medium permeability.

- p_A (SI unit: Pa) is the absolute pressure.

- ρ_l (SI unit: kg/m^3) is the liquid water density, defined as a function of the temperature.

- $\mathbf{g}_{lc}$ is the liquid water capillary flux, defined either from the capillary pressure gradient by a Darcy's law:

$$\mathbf{g}_{lc} = \rho_l \frac{\kappa_{rl}\kappa}{\mu_l} \nabla p_c$$

or by a diffusion equation using the relative humidity:

$$\mathbf{g}_{lc} = -D_w \frac{\partial w(\phi_w)}{\partial \phi_w} \nabla \phi_w$$

- D_w (SI unit: m^2/s) is the moisture diffusivity.

- p_c is the capillary pressure, that can be related to the relative humidity using Kelvin's law:

$$p_c = -\frac{RT\rho_l}{M_v} \ln(\phi_w)$$

- G (SI unit: $kg/(m^3 \cdot s)$) is a moisture source (or sink). See the Moisture Source node.

The moisture source due to evaporation may be obtained from the transport equation for vapor:

$$G_{evap} = \frac{\partial[\varepsilon_p \rho_g \omega_v(1-s_l)]}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w$$

The corresponding latent heat source is defined as:

$$Q_{evap} = L_v G_{evap}$$

where L_v is the latent heat of evaporation.

For a steady-state problem, the relative humidity does not change with time and the first term in the moisture transport equation disappears.

Theory for Moisture Transport in Building Materials

By considering that building materials are in general surrounded by an environment of constant atmospheric pressure and under a certain range of temperature and moisture

conditions, specific equations are proposed in Ref. 16 for moisture transport in this kind of materials.

Starting from the equation for the transport of moisture in porous media (using the diffusion model for capillary flux):

$$\frac{\partial w(\phi_w)}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w + \mathbf{u}_l \cdot \nabla \rho_l + \nabla \cdot \left(-D_w \frac{\partial w(\phi_w)}{\partial \phi_w} \nabla \phi_w\right) = G$$

let's make the assumption that there is no variation of the ambient total pressure (p_A=1 atm). In this case, by defining the gas and liquid velocities $\mathbf{u}_g$ and $\mathbf{u}_l$ as Darcy velocities, the convective fluxes for vapor and liquid water vanish:

$$\frac{\partial w(\phi_w)}{\partial t} + \nabla \cdot \left(\mathbf{g}_w - D_w \frac{\partial w \phi_w}{\partial \phi_w} \nabla \phi_w\right) = G$$

By introducing the moisture storage capacity, ξ, defined as

$$\xi = \frac{\partial w(\phi_w)}{\partial \phi_w}$$

the equation reads

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (\mathbf{g}_w - D_w \xi \nabla \phi_w) = G$$

Finally, the diffusive flux for vapor can be written as:

$$\mathbf{g}_w = -\rho_g D_{eff} \nabla \omega_v = -\rho_g D_{eff} \nabla \left[\frac{M_a M_v}{(x_a M_a + x_v M_v)^2} \frac{p_v}{p_A}\right]$$

By making the approximation that $x_a M_a + x_v M_v$ is constant, and that moist air is an ideal gas, we have:

$$\mathbf{g}_w = -\frac{M_v D_{eff}}{R_{H_2O}} \frac{M_a}{(x_a M_a + x_v M_v)^2} \nabla p_v$$

The vapor permeability coefficient is then defined as

$$\delta_p = \frac{M_v D_{eff}}{R_{H_2O}} \frac{M_a}{(x_a M_a + x_v M_v)^2}$$

The Moisture Transport in Building Materials Interface solves for the following equation derived from Ref. 16:

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (-\xi D_w \nabla \phi_w - \delta_p \nabla (\phi_w p_{sat}(T))) = G \tag{4-131}$$

This equation models the moisture transfer as the sum of the capillary moisture flux:

$$-D_w \nabla (w(\phi_w)) = -D_w \frac{\partial w}{\partial \phi_w} \nabla \phi_w = -\xi D_w \nabla \phi_w$$

and the vapor diffusion flux:

$$\delta_p \nabla p_v(T) = \delta_p \nabla (\phi_w p_{sat}(T))$$

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_w (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.

Theory for the Heat Transfer Multiphysics Couplings

In this section:

- Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces
- Theory for the Moisture Flow Interface
- Theory for the Thermoelectric Effect Interface
- Theory for the Building Materials Version of the Heat and Moisture Transport Interface
- Theory for the Moist Air Version of the Heat and Moisture Transport Interface
- Theory for the Moist Porous Media Version of the Heat and Moisture Transport Interface
- Theory for the Heat and Moisture Flow Interfaces
- Theory for the Electromagnetic Heating Interfaces
- Theory for the Thermal Stress Interface

Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces

The following points of the theory of Nonisothermal Flow and Conjugate Heat Transfer are discussed in this part:

- The Nonisothermal Flow and Conjugate Heat Transfer Equations
- Turbulent Nonisothermal Flow Theory
- Theory for the Nonisothermal 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.

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 Nonisothermal 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 Nonisothermal Flow and Conjugate Heat Transfer interfaces contain the fully compressible formulation of the continuity and momentum equations:

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

(4-132)

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \nabla \cdot \tau + \mathbf{F}$$

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:

$$\tau = \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}$$

- 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 4-18 by

$$\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T\right) + \nabla \cdot (\mathbf{q} + \mathbf{q_r}) = \alpha_p T\left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p\right) + \tau : \nabla \mathbf{u} + Q$$

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)

- $\mathbf{q_r}$ 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

$$Q_{vd} = \tau{:}\nabla \mathbf{u}$$

are not included by default because they are usually negligible. These terms can, however, be added by selecting corresponding check boxes in the Nonisothermal Flow feature.

The physics interface also supports heat transfer in solids (Equation 4-16):

$$\rho C_p\left(\frac{\partial T}{\partial t} + \mathbf{u}_{trans} \cdot \nabla T\right) + \nabla \cdot (\mathbf{q} + \mathbf{q_r}) = Q_{ted} + Q$$

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 Nonisothermal Flow Theory

TURBULENT NONISOTHERMAL 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

$$\frac{\partial}{\partial t}\left(\bar{\rho}\left(\tilde{E} + \frac{\tilde{u}_i \tilde{u}_i}{2}\right) + \frac{\overline{\rho u_i'' u_i''}}{2}\right) + \frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\left(\tilde{H} + \frac{\tilde{u}_i \tilde{u}_i}{2}\right) + \tilde{u}_j\frac{\overline{\rho u_i'' u_i''}}{2}\right) = \quad (4\text{-}133)$$

$$\frac{\partial}{\partial x_j}\left(-q_j - \overline{\rho u_j'' H''} + \overline{\tau_{ij} u_i''} - \frac{\overline{\rho u_j'' u_i'' u_i''}}{2}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}} - \overline{\rho u_i'' u_j''}))$$

where H is the enthalpy. The vector

$$q_j = -\lambda\frac{\partial T}{\partial x_j} \quad (4\text{-}134)$$

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*:

$$-\overline{\rho u_i'' u_j''} = \bar{\rho}\tau^{\mathrm{T}}_{ij} = \mu_{\mathrm{T}}\left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i}\right) - \frac{2}{3}\mu\frac{\partial \tilde{u}_k}{\partial x_k}\delta_{ij} - \frac{2}{3}\bar{\rho}k\delta_{ij} \quad (4\text{-}135)$$

where k is the turbulent kinetic energy, which in turn is defined by

$$\bar{\rho}k = \frac{1}{2}\overline{\rho u_i'' u_i''} \quad (4\text{-}136)$$

The correlation between u_j'' and H'' in Equation 4-133 is the turbulent transport of heat. It is modeled analogously to the laminar conductive heat flux

$$\overline{\rho u_j'' H''} = q_j^{\mathrm{T}} = -\lambda_{\mathrm{T}}\frac{\partial \tilde{T}}{\partial x_j} = -\frac{\mu_{\mathrm{T}} C_p}{\mathrm{Pr}_{\mathrm{T}}}\frac{\partial \tilde{T}}{\partial x_j} \quad (4\text{-}137)$$

The molecular diffusion term,

$$\overline{\tau_{ij}u_i''}$$

and turbulent transport term,

$$\overline{\frac{\rho u_j'' u_i'' u_i''}{2}}$$

are modeled by a generalization of the molecular diffusion and turbulent transport terms found in the incompressible k equation

$$\overline{\tau_{ij}u_i''} - \overline{\frac{\rho u_j'' u_i'' u_i''}{2}} = \left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j} \tag{4-138}$$

Inserting Equation 4-134, Equation 4-135, Equation 4-136, Equation 4-137 and Equation 4-138 into Equation 4-133 gives

$$\frac{\partial}{\partial t}\left(\bar{\rho}\left(\tilde{E} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k\right)\right) + \frac{\partial}{\partial x_j}\left(\bar{\rho}\tilde{u}_j\left(\tilde{H} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k\right)\right) = \tag{4-139}$$

$$\frac{\partial}{\partial x_j}\left(-q_j - q_j^{\mathrm{T}} + \left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}} + \bar{\rho}\tau_{ij}^{\mathrm{T}}))$$

The Favre average can also be applied to the momentum equation, which, using Equation 4-135, can be written

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{u}_i) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{u}_i) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}(\overline{\tau_{ij}} + \bar{\rho}\tau_{ij}^{\mathrm{T}}) \tag{4-140}$$

Taking the inner product between $\tilde{u}_i$ and Equation 4-140 results in an equation for the resolved kinetic energy, which can be subtracted from Equation 4-139 with the following result:

$$\frac{\partial}{\partial t}(\bar{\rho}(\tilde{E} + k)) + \frac{\partial}{\partial x_j}(\rho\tilde{u}_j(\tilde{E} + k)) = \tag{4-141}$$

$$-\bar{p}\frac{\partial \tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}\left(-q_j - q_j^{\mathrm{T}} + \left(\mu + \frac{\mu_T}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}} + \bar{\rho}\tau_{ij}^{\mathrm{T}}))$$

where the relation

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

has been used.

According to Wilcox (Ref. 27), 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 4-141:

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E}) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial \tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}(-\bar{q}_j - q_j^{\mathrm{T}}) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\overline{\tau_{ij}} + \bar{\rho}\tau_{ij}^{\mathrm{T}})) \qquad (4\text{-}142)$$

Larsson (Ref. 28) 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

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E}) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial \tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}\left((\lambda + \lambda_{\mathrm{T}})\frac{\partial \tilde{T}}{\partial x_j}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i\tilde{\tau}_{ij}^{\mathrm{tot}}) \qquad (4\text{-}143)$$

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 4-143 is completely analogous to the laminar energy equation of Equation 4-15 and can be expanded using the same theory to get the temperature equation similar to Equation 4-18 (see for example Ref. 28):

$$\bar{\rho}C_p\frac{\partial \tilde{T}}{\partial t} + \bar{\rho}C_p\tilde{u}_j\frac{\partial \tilde{T}}{\partial x_j} + \frac{\partial}{\partial x_j}\left(-(\lambda + \lambda_{\mathrm{T}})\frac{\partial \tilde{T}}{\partial x_j}\right) = -\frac{1}{\bar{\rho}}\left(\frac{\partial \bar{\rho}}{\partial \tilde{T}}\right)_p \tilde{T}\left(\frac{\partial \bar{p}}{\partial t} + \tilde{u}_j\frac{\partial \bar{p}}{\partial x_j}\right) + \tilde{\tau}_{ij}\frac{\partial \tilde{u}_i}{\partial x_j}$$

which is the temperature equation solved in the turbulent Nonisothermal 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. 29, 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

$$\mathrm{Pr_T} = \left(\frac{1}{2\mathrm{Pr_{T\infty}}} + \frac{0.3 C_p \mu_T}{\lambda \sqrt{\mathrm{Pr_{T\infty}}}} - \left(\frac{0.3 C_p \mu_T}{\lambda} \right)^2 \left(1 - \exp\left(-\frac{\lambda}{0.3 C_p \mu_T \sqrt{\mathrm{Pr_{T\infty}}}} \right) \right) \right)^{-1} \quad (4\text{-}144)$$

where the Prandtl number at infinity is $\mathrm{Pr_{T\infty}} = 0.85$ and λ is the conductivity.

Extended Kays-Crawford Weigand and others (Ref. 30) suggested an extension of Equation 4-144 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:

$$q_{\mathrm{wf}} = \frac{\rho C_p u_\tau (T_w - T_f)}{T^+}$$

where ρ is the fluid density, C_p is the fluid heat capacity, and u_τ is the friction velocity. T^+ is the dimensionless temperature and is given by (Ref. 31):

$$
T^+ = \begin{cases}
\mathrm{Pr}\delta_w^+ & \text{for } \delta_w^+ < \delta_{w1}^+ \\[2mm]
15\mathrm{Pr}^{2/3} - \dfrac{500}{\delta_w^{+2}} & \text{for } \delta_{w1}^+ \leq \delta_w^+ < \delta_{w2}^+ \\[2mm]
\dfrac{\mathrm{Pr}_T}{\kappa}\ln\delta_w^+ + \beta & \text{for } \delta_{w2}^+ \leq \delta_w^+
\end{cases}
$$

where in turn

$$
\delta_w^+ = \frac{\delta_w \rho \sqrt{C_\mu^{1/2} k}}{\mu} \qquad\qquad \delta_{w1}^+ = \frac{10}{\mathrm{Pr}^{1/3}}
$$

$$
\delta_{w2}^+ = 10\sqrt{10\frac{\kappa}{\mathrm{Pr}_T}} \qquad\qquad \mathrm{Pr} = \frac{C_p\mu}{\lambda}
$$

$$
\beta = 15\mathrm{Pr}^{2/3} - \frac{\mathrm{Pr}_T}{2\kappa}\left(1 + \ln\left(1000\frac{\kappa}{\mathrm{Pr}_T}\right)\right)
$$

λ is the thermal conductivity, and κ is the von Karman constant equal to 0.41.

The distance between the computational fluid domain and the wall, δ_w, is always $h_w/2$ for automatic wall treatment where h_w is the height of the mesh cell adjacent to the wall. $h_w/2$ is almost always very small compared to any geometrical quantity of interest, at least if a boundary layer mesh is used. For wall function, δ_w is at least $h_w/2$ and can be bigger if necessary to keep δ_w^+ higher than 11.06. 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 NONISOTHERMAL SCREEN BOUNDARY CONDITION

When the Nonisothermal Flow multiphysics coupling feature is active, the conditions that apply across a screen in isothermal flow are complemented by:

$$
[H_0]_-^+ = 0 \tag{4-145}
$$

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.

When the Nonisothermal 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 T_{avg} is defined by:

$$\left(\int_{\Gamma} \mathrm{down}(\mathbf{u} \cdot \mathbf{n} \rho C_p) dS \right) T_{\mathrm{avg}} = \int_{\Gamma} \mathrm{down}(\mathbf{u} \cdot \mathbf{n} \rho C_p T) dS$$

- If the direction is opposite to normal vector, the outlet temperature T_{avg} is defined by:

$$\left(\int_{\Gamma} \mathrm{up}(\mathbf{u} \cdot \mathbf{n} \rho C_p) dS \right) T_{\mathrm{avg}} = \int_{\Gamma} \mathrm{up}(\mathbf{u} \cdot \mathbf{n} \rho C_p T) dS$$

Theory for the Moisture Flow Interface

The following points of the theory of Moisture Flow are discussed in this part:

- The Moisture Flow Equations
- Stefan Flow at Walls
- Turbulent Moisture Flow Theory
- Theory for the Screen Boundary Condition in Moisture Flow
- Theory for the Interior Fan Boundary Condition in Moisture Flow

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 MOISTURE FLOW EQUATIONS

This module includes the Moisture Flow predefined multiphysics coupling to simulate systems in which the airflow depends on moisture content through its density and viscosity. It can be used in free and hygroscopic porous media containing moist air.

The density and viscosity of air might also vary due to temperature variations. See Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces for details on this other kind of dependency.

The Moisture Flow interface contains the fully compressible formulation of the continuity and momentum equations. For laminar flow they read:

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

$$\rho_g \frac{\partial \mathbf{u}}{\partial t} + \rho_g \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \nabla \cdot \tau + \mathbf{F}$$

(4-146)

where

- ρ_g is the moist air 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:

$$\tau = \mu_g (\nabla \mathbf{u} + (\nabla \mathbf{u})^{\mathrm{T}}) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}$$

- μ_g is the moist air dynamic viscosity (SI unit: Pa·s)
- $\mathbf{F}$ is the body force vector (SI unit: N/m^3)

It also solves the equation for moisture transport in air, given in Equation 4-128 by

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w = G$$

where

- M_v is the molar mass of water vapor (SI unit: kg/mol)
- c_v is the vapor concentration (SI unit: mol/m^3)
- $\mathbf{g}_w$ is the moisture flux by diffusion (SI unit: kg/(m^2·s)):

$$\mathbf{g}_w = -M_v D \nabla c_v$$

- D (SI unit: m^2/s) is the vapor diffusion coefficient in air.
- G contains moisture sources (SI unit: kg/(m^3·s))

When the spatial and temporal variations of the vapor concentration induce significant variations of the moist air density, the moisture content variation is expressed through the transport of vapor mass fraction, ω_v, defined as:

$$\omega_v = \frac{M_v c_v}{\rho_g}$$

The equation for moisture transport in air is then expressed as follows:

$$\rho_g \frac{\partial \omega_v}{\partial t} + \rho_g \mathbf{u} \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w \ = \ G$$

with

$$\mathbf{g}_w = -\rho_g D \nabla \omega_v$$

STEFAN FLOW AT WALLS

Surface reactions like evaporation and condensation result in a net moisture flux $\mathbf{g}_w$ between the boundary and the domain. The resulting effective velocity at the boundary is the Stefan velocity u_{Stefan}, defined as

$$\rho_g u_{\text{Stefan}} \ = \ \mathbf{n} \cdot \mathbf{g}_w$$

The Moisture Flow interface allows to use the Stefan velocity as a leakage velocity at walls in the fluid flow equations.

TURBULENT MOISTURE FLOW THEORY

The Favre average $\tilde{c}_v$ of the vapor concentration c_v is defined by

$$\tilde{c}_v \ = \ \frac{\overline{\rho_g c_v}}{\overline{\rho_g}}$$

where the bar denotes the usual Reynolds average. The full field can be decomposed as the sum of the Favre average and the Favre fluctuation:

$$c_v \ = \ \tilde{c}_v + c_v{''}$$

By applying this decomposition and averaging the moisture transport equations, unclosed terms are introduced in the equations. Among them, the turbulent moisture transport flux contains the velocity and vapor concentration fluctuations:

$$u_j{''} \frac{\partial c_v{''}}{\partial x_j}$$

This flux is modeled using a gradient based assumption, where the additional transport is related to the turbulent viscosity ν_T through the turbulent Schmidt number Sc_T. The turbulent diffusivity D_T is defined by

$$D_T \ = \ \frac{\nu_T}{Sc_T}$$

The following moisture transport equation is solved:

$$M_v \frac{\partial \tilde{c}_v}{\partial t} + M_v \tilde{u}_j \frac{\partial \tilde{c}_v}{\partial x_j} + M_v \frac{\partial}{\partial x_j}\left(-(D + D_T)\frac{\partial \tilde{c}_v}{\partial x_j}\right) = G \tag{4-147}$$

Kays-Crawford Model for Turbulent Diffusivity

By analogy with turbulent heat transfer, the turbulent Schmidt number Sc_T is given by (Ref. 29)

$$Sc_T = \left(\frac{1}{2Sc_{T\infty}} + \frac{0.3\mu_T Sc}{\mu\sqrt{Sc_{T\infty}}} - \left(0.3\frac{\mu_T Sc}{\mu}\right)^2\left(1 - \exp\left(-\frac{\mu}{0.3\mu_T Sc\sqrt{Sc_{T\infty}}}\right)\right)\right)^{-1} \tag{4-148}$$

where the Schmidt number at infinity is $Sc_{T\infty} = 0.85$ and the Schmidt number is defined as

$$Sc = \frac{\nu}{D}$$

where ν is the kinematic viscosity.

Moisture Transport 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 relative humidity fields. This gap is often ignored when the computational geometry is drawn.

Assuming that the turbulent heat and moisture transfer in the near-wall region are analogous, the same type of wall functions used for the temperature (Ref. 31) is also applicable for the moisture transport. The moisture transfer wall function is formulated as a function of the turbulent Schmidt number, instead of the corresponding Prandtl number.

The moisture flux at the lift-off position between the air with vapor mass fraction $\omega_{v,a}$ and a wall with vapor mass fraction $\omega_{v,w}$, is:

$$g_{wf} = \frac{\rho_g u_\tau (\omega_{v,\,w} - \omega_{v,\,a})}{\omega_w^+}$$

where u_τ is the friction velocity, and ω_w^+ is the dimensionless mass fraction given by (Ref. 31):

$$\phi_w^+ = \begin{cases} Sc\delta_w^+ & \text{for } \delta_w^+ < \delta_{w1}^+ \\[2mm] 15Sc^{2/3} - \dfrac{500}{\delta_w^{+2}} & \text{for } \delta_{w1}^+ \le \delta_w^+ < \delta_{w2}^+ \\[2mm] \dfrac{Sc_T}{\kappa}\ln\delta_w^+ + \beta & \text{for } \delta_{w2}^+ \le \delta_w^+ \end{cases}$$

where in turn

$$\delta_w^+ = \frac{\delta_w \rho \sqrt{C_\mu^{1/2} k}}{\mu} \qquad\qquad \delta_{w1}^+ = \frac{10}{Sc^{1/3}}$$

$$\delta_{w2}^+ = 10\sqrt{10\frac{\kappa}{Sc_T}} \qquad\qquad Sc = \frac{\mu}{\rho D}$$

$$\beta = 15Sc^{2/3} - \frac{Sc_T}{2\kappa}\left(1 + \ln\left(1000\frac{\kappa}{Sc_T}\right)\right)$$

where κ is the von Karman constant equal to 0.41, C_μ is a turbulence modeling constant, and k is the turbulent kinetic energy.

The distance between the computational fluid domain and the wall, δ_w, is always $h_w/2$ for automatic wall treatment where h_w is the height of the mesh cell adjacent to the wall. $h_w/2$ is almost always very small compared to any geometrical quantity of interest, at least if a boundary layer mesh is used. For wall function, δ_w is at least $h_w/2$ and can be bigger if necessary to keep δ_w^+ higher than 11.06. 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 SCREEN BOUNDARY CONDITION IN MOISTURE FLOW

When the Moisture Flow multiphysics coupling feature is active, the conditions that apply across a screen in moisture flow are complemented by:

$$[c_v]_-^+ = 0 \tag{4-149}$$

to ensure mass conservation.

> - 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 IN MOISTURE FLOW

When the Moisture 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 vapor concentration c_{avg} is defined by:

$$\left(\int_{\Gamma} \mathrm{down}(\rho_g \mathbf{u} \cdot \mathbf{n}) dS \right) c_{avg} = \int_{\Gamma} \mathrm{down}(\rho_g \mathbf{u} \cdot \mathbf{n} c_v) dS$$

- If the direction is opposite to normal vector, the outlet vapor concentration c_{avg} is defined by:

$$\left(\int_{\Gamma} \mathrm{up}(\rho_g \mathbf{u} \cdot \mathbf{n}) dS \right) c_{avg} = \int_{\Gamma} \mathrm{up}(\rho_g \mathbf{u} \cdot \mathbf{n} c_v) dS$$

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 **q**, defined by

$$\mathbf{q} = -k\nabla T + P\mathbf{J} \qquad (4\text{-}150)$$

• Electric current density **J**, defined by

$$\mathbf{J} = -\sigma(\nabla V + S\nabla T) \qquad (4\text{-}151)$$

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 **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 4-150 and Equation 4-151, 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

$$\mathbf{J}_{Se} = -\sigma S \nabla T$$

This formulation corresponds directly to Equation 4-151 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

$$Q_{Pe} = -P \nabla \cdot \mathbf{J}$$

This contribution is obtained by developing the divergence of **q** term in the heat equation when **q** is defined following Equation 4-150.

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

$$Q_{Th} = -\mu_{Th} \mathbf{J} \cdot \nabla T$$

This contribution is obtained again by developing the divergence of the q term in the heat equation when **q** is defined following Equation 4-150. This time consider the term $-T\mathbf{J} \cdot \nabla S$. Assuming that S is function of T, then:

$$-T\mathbf{J} \cdot \nabla S = -T\frac{dS}{dT}\mathbf{J} \cdot \nabla T = -\mu_{Th}\mathbf{J} \cdot \nabla T$$

Theory for the Building Materials Version of the Heat and Moisture Transport Interface

The Building Materials version of the Heat and Moisture multiphysics coupling implements the following equations for heat and moisture transport, derived from Ref. 16:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot (-k_{\text{eff}} \nabla T - L_{\text{v}} \delta_{\text{p}} \nabla(\phi_{\text{w}} p_{\text{sat}})) = Q \qquad (4\text{-}152)$$

$$\xi \frac{\partial \phi_{\text{w}}}{\partial t} + \nabla \cdot (-\xi D_{\text{w}} \nabla \phi_{\text{w}} - \delta_{\text{p}} \nabla(\phi_{\text{w}} p_{\text{sat}})) = G \qquad (4\text{-}153)$$

See Theory for Heat Transfer in Building Materials and Theory for Moisture Transport in Building Materials for more details about the notations.

The heat transfer and moisture transport equations are coupled through:

- the definition of the effective thermal properties accounting for moisture content,
- the definition of the latent heat source from the evaporation source in domains,
- the definition of the vapor saturation in function of the temperature.

Theory for the Moist Air Version of the Heat and Moisture Transport Interface

The Moist Air version of the Heat and Moisture multiphysics coupling implements the following equations in domains for heat and moisture transport:

$$\rho_{\text{g}} C_p \frac{\partial T}{\partial t} + \rho_{\text{g}} C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \qquad (4\text{-}154)$$

$$\mathbf{q} = -k \nabla T \qquad (4\text{-}155)$$

$$Q = -(C_{p,\text{v}} - C_{p,\text{a}}) \mathbf{g}_{\text{w}} \qquad (4\text{-}156)$$

$$M_{\text{v}} \frac{\partial c_{\text{v}}}{\partial t} + M_{\text{v}} \mathbf{u} \cdot \nabla c_{\text{v}} + \nabla \cdot \mathbf{g}_{\text{w}} = G \qquad (4\text{-}157)$$

$$\mathbf{g}_{\text{w}} = -M_{\text{v}} D \nabla c_{\text{v}} \qquad (4\text{-}158)$$

$$c_{\text{v}} = \phi_{\text{w}} c_{\text{sat}} \qquad (4\text{-}159)$$

See Theory for Heat Transfer in Moist Air and Theory for Moisture Transport in Air for more details about the notations.

In addition, on boundaries where evaporation or condensation takes place, the following equation is solved:

$$q_{\text{evap}} = L_{\text{v}} g_{\text{evap}} \qquad (4\text{-}160)$$

The heat transfer and moisture transport equations are coupled through:

- the definition of the effective thermal properties accounting for vapor concentration in moist air,
- the definition of the diffusive flux of enthalpy (Equation 4-156),
- the definition of the latent heat source from the evaporation source on surfaces (Equation 4-160),
- the definition of the vapor saturation in function of the temperature (Equation 4-159).

Theory for the Moist Porous Media Version of the Heat and Moisture Transport Interface

The Moist Porous Media version of the Heat and Moisture multiphysics coupling implements the following equations in domains for heat and moisture transport:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + (\rho_g C_{p,\,g} \mathbf{u}_g + \rho_l C_{p,\,l} \mathbf{u}_l) \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + Q_{\text{evap}} \qquad (4\text{-}161)$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \qquad (4\text{-}162)$$

$$Q = -[(C_{p,\,\text{v}} - C_{p,\,\text{a}}) \mathbf{g}_{\text{w}} + C_{p,\,l} \mathbf{g}_{\text{lc}}] \qquad (4\text{-}163)$$

$$Q_{\text{evap}} = L_{\text{v}} G_{\text{evap}} \qquad (4\text{-}164)$$

$$\frac{\partial w(\phi_{\text{w}})}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_{\text{v}} + \nabla \cdot \mathbf{g}_{\text{w}} + \mathbf{u}_l \cdot \nabla \rho_l + \nabla \cdot \mathbf{g}_{\text{lc}} = G \qquad (4\text{-}165)$$

$$\mathbf{g}_{\text{w}} = -\rho_g D_{\text{eff}} \nabla \omega_{\text{v}} \qquad (4\text{-}166)$$

$$G_{\text{evap}} = \frac{\partial [\varepsilon_p \rho_g \omega_{\text{v}} (1 - s_l)]}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_{\text{v}} + \nabla \cdot \mathbf{g}_{\text{w}} \qquad (4\text{-}167)$$

See Theory for Heat Transfer in Moist Porous Media and Theory for Moisture Transport in Porous Media for more details about the notations.

The heat transfer and moisture transport equations are coupled through:

- the definition of the effective thermal properties accounting for moisture content in liquid and gas phases,
- the liquid water saturation and velocity field used to defined the convective heat flux,
- the diffusive flux of enthalpy in moist air and the liquid capillary flux (Equation 4-163)
- the definition of the latent heat source from the evaporation source in domains (Equation 4-164),
- the definition of the vapor saturation in function of the temperature.

Theory for the Heat and Moisture Flow Interfaces

The theory for the equations of heat transfer and moisture transport by laminar and turbulent air flows can be found in Theory for the Moisture Flow Interface and Theory for the Moist Air Version of the Heat and Moisture Transport Interface.

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 the multiphysics coupling feature Electromagnetic Heating, which adds weak contributions due to resistive losses in the domains and boundaries, and shares the temperature variable with the electromagnetics interfaces. 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*.

Theory for the Thermal Stress Interface

In the Structural Mechanics interfaces, two multiphysics interfaces handle thermal stress:

• The Thermal Stress, Solid 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

Theory for the Thermal Contact Feature

The Thermal Contact feature node 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:

$$-\mathbf{n}_d \cdot (-k_d \nabla T_d) = -h(T_u - T_d) + rQ_b$$

$$-\mathbf{n}_u \cdot (-k_u \nabla T_u) = -h(T_d - T_u) + (1-r)Q_b$$

At a microscopic level, contact is made at a finite number of spots as in Figure 4-23.
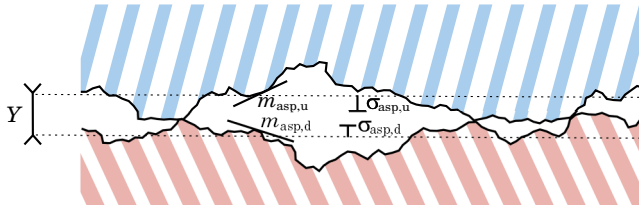
Figure 4-23: Contacting surfaces at the microscopic level.

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:

$$h = h_c + h_g + h_r$$

SURFACE ASPERITIES

The microscopic surface asperities are characterized by the average height $\sigma_{u,\,asp}$ and $\sigma_{d,\,asp}$ and the average slope $m_{u,\,asp}$ and $m_{d,\,asp}$. The RMS values σ_{asp} and m_{asp} are (4.16 in Ref. 1):

$$\sigma_{asp} = \sqrt{\sigma_{u,\,asp}^2 + \sigma_{d,\,asp}^2} \qquad m_{asp} = \sqrt{m_{u,\,asp}^2 + m_{d,\,asp}^2}$$

CONSTRICTION CONDUCTANCE

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:

$$h_c = 1.25 k_{\text{contact}} \frac{m_{\text{asp}}}{\sigma_{\text{asp}}} \left(\frac{p}{H_c}\right)^{0.95}$$

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:

$$k_{\text{contact}} = \frac{2 k_u k_d}{k_u + k_d}$$

📄 | When k_u (resp. k_d) is not isotropic, it is replaced by its normal conductivity $\mathbf{n}^{\text{T}} k_u \mathbf{n}$ (resp. $\mathbf{n}^{\text{T}} k_d \mathbf{n}$).

The relative pressure p/H_c can be evaluated by specifying H_c directly or using the following relation (4.16.1 in Ref. 1) for the relative pressure using c_1 and c_2, the Vickers correlation coefficient and size index:

$$\frac{p}{H_c} = \left(\frac{p}{c_1 \left(1.62 \frac{\sigma_{\text{asp}}}{\sigma_0} m_{\text{asp}}\right)^{c_2}} \right)^{\frac{1}{(1 + 0.071 c_2)}}$$

The coefficients c_1 and c_2 are the Vickers correlation coefficient and size index, respectively, and σ_0 is equal to 1 μm. For materials with 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. 1):

$$\frac{c_1}{H_0} = 4.0 - 5.77 \frac{H_B}{H_0} + 4.0 \left(\frac{H_B}{H_0}\right)^2 - 0.61 \left(\frac{H_B}{H_0}\right)^3$$

$$c_2 = -0.37 + 0.442\frac{H_{\mathrm{B}}}{c_1}$$

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:

$$h_{\mathrm{c}} = 1.54k_{\mathrm{contact}}\frac{m_{\mathrm{asp}}}{\sigma_{\mathrm{asp}}}\left(\frac{\sqrt{2}p}{mE_{\mathrm{contact}}}\right)^{0.94}$$

Here, E_{contact} is an effective Young's modulus for the contact interface, satisfying (4.16.3 in Ref. 1):

$$\frac{1}{E_{\mathrm{contact}}} = \frac{1 - v_{\mathrm{u}}^2}{E_{\mathrm{u}}} + \frac{1 - v_{\mathrm{d}}^2}{E_{\mathrm{d}}}$$

where E_{u} and E_{d} are the Young's moduli of the two contacting surfaces and v_{u} and v_{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:

$$h_{\mathrm{g}} = \frac{k_{\mathrm{g}}}{Y + M_{\mathrm{g}}}$$

Here k_{g} is the gas conductivity, Y denotes the mean separation thickness (see Figure 4-23), and M_{g} is the gas parameter equal to:

$$M_{\mathrm{g}} = \alpha\beta\Lambda \qquad \Lambda = \frac{k_{\mathrm{B}}T_{\mathrm{g}}}{\sqrt{2}\pi D^2 p_{\mathrm{g}}}$$

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, 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. 2) 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, $\mathbf{n}^T k_d \mathbf{n}$ (resp. $\mathbf{n}^T k_u \mathbf{n}$) replaces k_d (resp. k_u).

⊕	Thermal Contact

▥	*Contact Switch*: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/contact_switch**

REFERENCES

1. A. Bejan and others, *Heat Transfer Handbook*, John Wiley & Sons, 2003.

2. F. Charron, "Partage de la chaleur entre deux corps frottants," *Publication Scientifique et Technique du Ministère de l'Air*, no. 182, 1943.

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 4-24 shows examples of possible situations in which this type of geometry reduction can be applied.
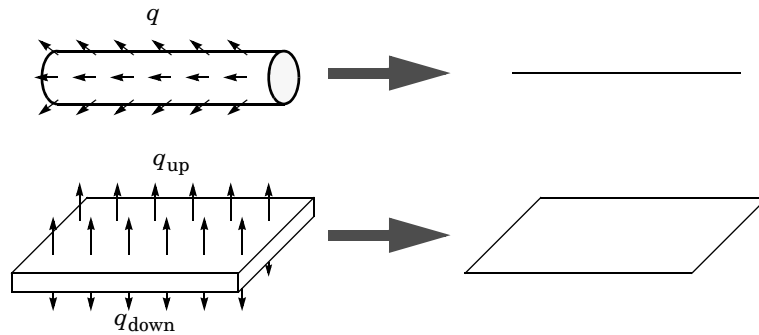
Figure 4-24: Geometry reduction from 3D to 1D (top) and from 3D to 2D (bottom).

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 4-24 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 4-16, and in fluids, Equation 4-18, are replaced by:

$$d_z \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = d_z Q + q_0 \tag{4-168}$$

$$d_z \rho C_p \frac{\partial T}{\partial t} + \rho C_p d_z \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = d_z Q + q_0 \tag{4-169}$$

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 6-15 is replaced by

$$(2\pi r d_z)\rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = (2\pi r d_z)Q + q_0 \tag{4-170}$$

where d_z is the thickness of the domain in the z direction. The equation for heat transfer in fluids, Equation 6-5, is replaced by

$$(2\pi r d_z)\rho C_p \frac{\partial T}{\partial t} + (2\pi r d_z)\rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = (2\pi r d_z)Q + q_0 \tag{4-171}$$

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 6-15 is replaced by

$$A_c \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = A_c Q + q_0 \tag{4-172}$$

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 6-5, is replaced by

$$A_c \rho C_p \frac{\partial T}{\partial t} + A_c \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = A_c Q + q_0 \tag{4-173}$$

Here, the conductive heat flux, $\mathbf{q}$, becomes

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

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:

$$q_0 = P_c q_{0, z}$$

where $q_{0, z}$ is the heat flux density distributed along the cross section perimeter.

Convective Heat Transfer Correlations

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 Nonisothermal 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

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\text{ext}} - T)$$

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
- Using the Heat and Mass Transfer Analogy for the Evaluation of Moisture Transfer Coefficients

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 4-25.
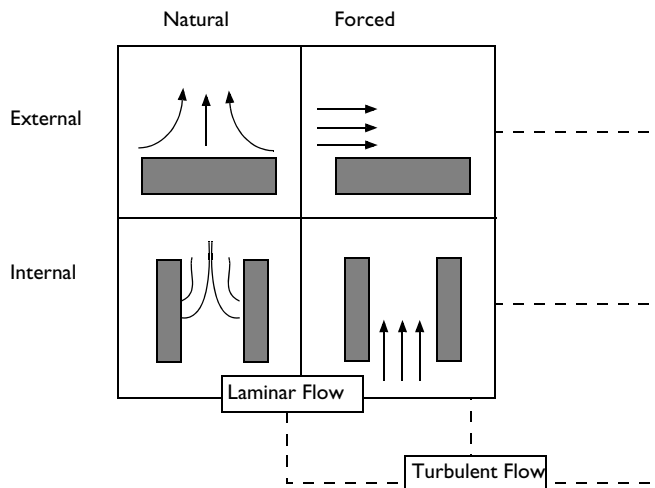
Figure 4-25: The eight categories of convective heat flux.

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, $\mathrm{Re}_L = \rho U L / \mu$
- The Prandtl number, $\mathrm{Pr} = \mu C_p / k$
- The Rayleigh number, $\mathrm{Ra}_L = \mathrm{Gr}_L \mathrm{Pr}$

where

- h is the heat transfer coefficient (SI unit: $\mathrm{W}/(\mathrm{m}^2{\cdot}\mathrm{K})$)
- 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/s^2)
- k is the thermal conductivity of the fluid (SI unit: $\mathrm{W}/(\mathrm{m}{\cdot}\mathrm{K})$)
- ρ is the fluid density (SI unit: kg/m^3)
- 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: $\mathrm{J}/(\mathrm{kg}{\cdot}\mathrm{K})$)

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 density scale (densities difference).

The Grashof number, Gr, is defined as:

$$\mathrm{Gr}_L = \frac{g\rho(\rho_{\mathrm{ext}} - \rho_s)L^3}{\mu^2}$$

where g is the acceleration of gravity, ρ_s denotes the density of the hot surface, ρ_{ext} equals the free stream density, L is the length scale, μ represents the fluid's dynamic viscosity, and ρ its density.

This expression of the Grashof number accounts for the all the density variations in the fluid. This applies to moist air, for which the density depends on temperature and vapor concentration.

When the density depends on temperature only, in dry air for example, the Grashof number is well approximated by using the fluid's coefficient of thermal expansion α_p (SI unit: 1/K):

$$\mathrm{Gr}_L = \frac{g\alpha_p(T_s - T_{\text{ext}})L^3}{(\mu/\rho)^2}$$

where g is the acceleration of gravity, T_s denotes the temperature of the hot surface, T_{ext} equals the free stream temperature, 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: $\mathrm{Ra} = \mathrm{Pr}\,\mathrm{Gr}$. 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.
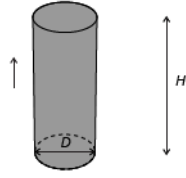
VERTICAL WALL
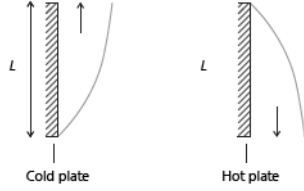
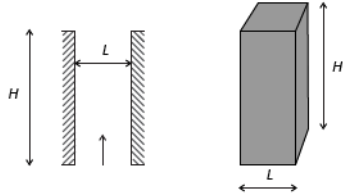Figure 4-26: *Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a vertical wall.*

The following correlations correspond to equations 9.26 and 9.27 in Ref. 21:

$$h = \begin{cases} \dfrac{k}{L}\left(0.68 + \dfrac{0.67 \mathrm{Ra}_L^{1/4}}{\left(1 + \left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{4/9}} \right) & \text{if } \mathrm{Ra}_L \leq 10^9 \\[4ex] \dfrac{k}{L}\left(0.825 + \dfrac{0.387 \mathrm{Ra}_L^{1/6}}{\left(1 + \left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{8/27}} \right)^2 & \text{if } \mathrm{Ra}_L > 10^9 \end{cases} \tag{4-174}$$

where the height of the wall, L, is a correlation input.

The Rayleigh number is defined as:

$$\mathrm{Ra}_L = \frac{g\alpha_p\rho^2 C_p |T - T_{\text{ext}}| L^3}{k\mu} \tag{4-175}$$

if the density depends on temperature only, or as:

$$\mathrm{Ra}_L = \frac{g\rho C_p |\rho_{\text{ext}} - \rho_{\text{s}}| L^3}{k\mu} \tag{4-176}$$

if the density depends on other parameters.

All material properties are evaluated at $(T + T_{\text{ext}})/2$, except ρ_{s}, which is evaluated at the wall temperature, T, and g is the acceleration of gravity equal to 9.81 m/s^2. This

correlation is valid for $10^4 \leq \mathrm{Ra}_L \leq 10^{13}$. The laminar-turbulent transition at $\mathrm{Ra}_L = 10^9$ is handled by the use of a smoothed Heaviside function with a transition of size 10^8, which corresponds to 10% of the threshold value.
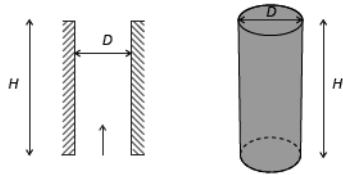
INCLINED WALL
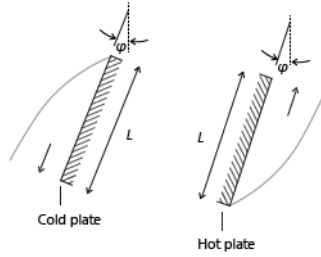
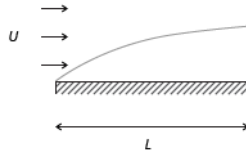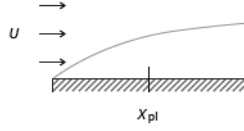*Figure 4-27: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on an inclined wall.*

The following correlations correspond to equations 9.26 and 9.27 in Ref. 21 (the same as for a vertical wall):

$$h = \begin{cases} \dfrac{k}{L}\left(0.68 + \dfrac{0.67(\cos\varphi\,\mathrm{Ra}_L)^{1/4}}{\left(1 + \left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{4/9}} \right) & \text{if } \mathrm{Ra}_L \leq 10^9 \\[4em] \dfrac{k}{L}\left(0.825 + \dfrac{0.387\,\mathrm{Ra}_L^{1/6}}{\left(1 + \left(\dfrac{0.492k}{\mu C_p}\right)^{9/16}\right)^{8/27}} \right)^2 & \text{if } \mathrm{Ra}_L > 10^9 \end{cases}$$

(4-177)

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; $\varphi = 0$ for vertical walls). These correlations are valid for $-60° < \varphi < 60°$ and $10^4 \leq \mathrm{Ra}_L \leq 10^{13}$.

The definition of the Raleigh number, Ra_L, is analogous to the one for vertical walls and is given by the following:

$$\mathrm{Ra}_L = \frac{g\alpha_p\rho^2 C_p |T - T_{\text{ext}}| L^3}{k\mu}$$

(4-178)

if the density depends on temperature only, or as:

$$\text{Ra}_L = \frac{g\rho C_p |\rho_{ext} - \rho_s| L^3}{k\mu} \tag{4-179}$$

if the density depends on other parameters.

g denotes the gravitational acceleration, equal to 9.81 m/s^2.

For turbulent flow, 1 is used instead of $\cos \varphi$ in the expression for h, because this gives better accuracy (see Ref. 41).

According to Ref. 21, 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. 41). Unfortunately, little data is available about transition. There is some data available in Ref. 41 but this data is only approximate, 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

$$\text{Ra}_L > 10^9$$

All material properties are evaluated at $(T + T_{ext})/2$, except ρ_s which is evaluated at the wall temperature, T. The laminar-turbulent transition at $\text{Ra}_L = 10^9$ is handled by the use of a smoothed Heaviside function with a transition of size 10^8, which corresponds to 10% of the threshold value.
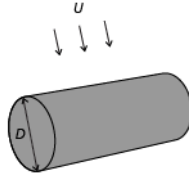
HORIZONTAL PLATE, UPSIDE
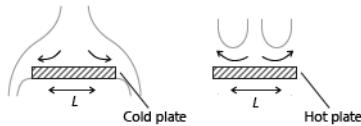
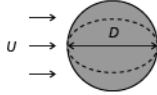*Figure 4-28: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on the top surface of an horizontal plate.*

The following correlations correspond to equations 9.30–9.32 in Ref. 21 but can also be found as equations 7.77 and 7.78 in Ref. 41.

If $\rho < \rho_{ext}$, then

$$h = \begin{cases} \dfrac{k}{L}0.54\mathrm{Ra}_L^{1/4} & \text{if } 10^4 \leq \mathrm{Ra}_L \leq 10^7 \\[2mm] \dfrac{k}{L}0.15\mathrm{Ra}_L^{1/3} & \text{if } 10^7 \leq \mathrm{Ra}_L \leq 10^{11} \end{cases} \tag{4-180}$$

while if $\rho \geq \rho_{ext}$, then

$$h = \frac{k}{L}0.27\mathrm{Ra}_L^{1/4} \text{ if } 10^5 \leq \mathrm{Ra}_L \leq 10^{10} \tag{4-181}$$

Ra_L is given by Equation 4-175 or Equation 4-176, and L, the characteristic length (defined as area/perimeter, see Ref. 41) is a correlation input. The material data are evaluated at $(T + T_{ext})/2$, except ρ_s which is evaluated at the wall temperature, T.

When the density depends only on temperature, the conditions $\rho < \rho_{ext}$ and $\rho \geq \rho_{ext}$ can be replaced by $T > T_{ext}$ and $T \leq T_{ext}$ respectively.

The laminar-turbulent transition at $\mathrm{Ra}_L = 10^7$ is handled by the use of a smoothed Heaviside function with a transition of size 10^6, which corresponds to 10% of the threshold value.

HORIZONTAL PLATE, DOWNSIDE
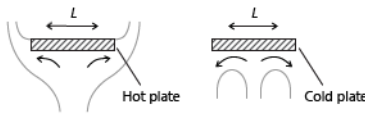
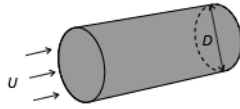*Figure 4-29: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on the bottom surface of an horizontal plate.*

Equation 4-180 is used when $\rho \geq \rho_{ext}$ (or $T \leq T_{ext}$) and Equation 4-181 is used when $\rho < \rho_{ext}$ (or $T > T_{ext}$). Otherwise it is the same implementation as for Horizontal Plate, Upside.

LONG HORIZONTAL CYLINDER
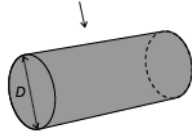
Figure 4-30: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a long horizontal cylinder.

The following correlations correspond to equations 9.34 in Ref. 21. It is validated for $Ra_D \leq 10^{12}$.

$$h = \frac{k}{D}\left(0.6 + \frac{0.387 Ra_D^{1/6}}{\left(1 + \left(\frac{0.559}{Pr}\right)^{9/16}\right)^{8/27}}\right)^2 \tag{4-182}$$

Here D is the cylinder diameter and Ra_D is given by

$$Ra_D = \frac{g\alpha_p\rho^2 C_p |T - T_{ext}|D^3}{k\mu}$$

if the density depends on temperature only, or as:

$$Ra_D = \frac{g\rho C_p |\rho_{ext} - \rho_s|D^3}{k\mu}$$

if the density depends on other parameters.

The material data are evaluated at $(T + T_{ext})/2$, except ρ_s which is evaluated at the wall temperature, T.

SPHERE
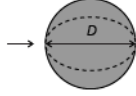
Figure 4-31: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a sphere.

The following correlations correspond to equation 9.35 in Ref. 21. It is validated for $\mathrm{Ra}_D \leq 10^{11}$ and $\mathrm{Pr} \geq 0.7$.

$$h = \frac{k}{D}\left(2 + \frac{0.589\mathrm{Ra}_D^{1/4}}{\left(1 + \left(\frac{0.469}{\mathrm{Pr}}\right)^{9/16}\right)^{4/9}}\right) \tag{4-183}$$

Here D is the cylinder diameter and Ra_D is given by

$$\mathrm{Ra}_D = \frac{g\alpha_p\rho^2 C_p|T - T_{\mathrm{ext}}|D^3}{k\mu}$$

if the density depends on temperature only, or as:

$$\mathrm{Ra}_D = \frac{g\rho C_p|\rho_{\mathrm{ext}} - \rho_s|D^3}{k\mu}$$

if the density depends on other parameters.

The material data are evaluated at $(T + T_{\mathrm{ext}})/2$, except ρ_s which is evaluated at the wall temperature, T.

VERTICAL THIN CYLINDER

Figure 4-32: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a vertical thin cylinder.

The following correlation corresponds to equation 7.83 in Ref. 41. It evaluates the heat transfer coefficient on the lateral surface of the thin cylinder. The horizontal disks (top and bottom) should be treated as horizontal plates.

For the lateral surface, the heat transfer coefficient reads:

$$h = \frac{k}{H}\left(\frac{4}{3}\left(\frac{7\mathrm{Ra}_H\mathrm{Pr}}{5(20+21\mathrm{Pr})}\right)^{1/4} + \frac{4(272+315\mathrm{Pr})H}{35(64+63\mathrm{Pr})D}\right)$$

where D is the cylinder diameter, H is the cylinder height, and Ra_H is given by

$$\mathrm{Ra}_H = \frac{g\alpha_p|T-T_{\mathrm{ext}}|H^3}{k_\mu}$$

if the density depends on temperature only, or as:

$$\mathrm{Ra}_H = \frac{g\rho C_p|\rho_{\mathrm{ext}}-\rho_s|H^3}{k\mu}$$

if the density depends on other parameters.

The material data are evaluated at $(T+T_{\mathrm{ext}})/2$, except ρ_s which is evaluated at the wall temperature, T.

If the thermal boundary layer thickness δ_T is much smaller than the cylinder diameter D, the curvature does not play any significant role, and vertical wall correlations should be used. In practice, the $(\delta_T < D)$ criterion requires

$$\frac{D}{H} > \mathrm{Ra}_H^{-\frac{1}{4}}$$

NARROW CHIMNEY, PARALLEL PLATES

Figure 4-33: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection in a parallel-plate narrow chimney.

If $\mathrm{Ra}_L < H/L$, then

$$h = \frac{k}{H}\frac{1}{24}\mathrm{Ra}_L \tag{4-184}$$

where the plate distance, L, and the chimney height, H, are correlation inputs (equation 7.96 in Ref. 41). Ra_L is given by Equation 4-175 or Equation 4-176. The material data are evaluated at $(T + T_{\mathrm{ext}})/2$.

NARROW CHIMNEY, CIRCULAR TUBE

Figure 4-34: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection in a circular narrow chimney.

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. 41 with $D_h = D$). Ra_D is given by Equation 4-175 or Equation 4-176 with L replaced by D. The material data are evaluated at $(T + T_{ext})/2$.

Heat Transfer Coefficients — External Forced Convection

PLATE, AVERAGED TRANSFER COEFFICIENT

Figure 4-35: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on an horizontal plate.

This correlation is a combination of equations 7.33 and 7.41 in Ref. 21:

$$
h = \begin{cases}
2\dfrac{k}{L}\dfrac{0.3387\mathrm{Pr}^{1/3}\mathrm{Re}_L^{1/2}}{\left(1+\left(\dfrac{0.0468}{\mathrm{Pr}}\right)^{2/3}\right)^{1/4}} & \text{if } \mathrm{Re}_L \leq 5\cdot 10^5 \\[4mm]
2\dfrac{k}{L}\mathrm{Pr}^{1/3}(0.037\mathrm{Re}_L^{4/5}-871) & \text{if } \mathrm{Re}_L > 5\cdot 10^5
\end{cases}
\tag{4-185}
$$

where $\mathrm{Pr} = \mu C_p/k$ and $\mathrm{Re}_L = \rho UL/\mu$. The plate length, L, and the exterior velocity, U, are correlation inputs. The material data are evaluated at $(T + T_{ext})/2$.

The laminar-turbulent transition at $\mathrm{Re}_L = 5.10^5$ is handled by the use of a smoothed Heaviside function with a transition of size 5.10^4, which corresponds to 10% of the threshold value.

PLATE, LOCAL TRANSFER COEFFICIENT

Figure 4-36: Schematic representation of geometry and parameters for the local heat transfer coefficient correlation applied to forced convection on an horizontal plate.

This correlation corresponds to equations 5.79b and 5.131' in Ref. 41:

$$
h = \begin{cases}
\dfrac{k}{x}0.332\mathrm{Pr}^{1/3}\mathrm{Re}_x^{1/2} & \text{if } \mathrm{Re}_x \leq 5 \cdot 10^5 \\[2ex]
\dfrac{k}{x}0.0296\mathrm{Pr}^{1/3}\mathrm{Re}_x^{4/5} & \text{if } \mathrm{Re}_x > 5 \cdot 10^5
\end{cases}
\tag{4-186}
$$

where $\mathrm{Pr} = \mu C_p/k$ and $\mathrm{Re}_x = \rho U x/\mu$. The correlation inputs are x, the position along the plate, and U, the exterior velocity. The material data are evaluated at $(T + T_{\text{ext}})/2$. To avoid division by zero when the position along the plate is located at the origin point $(x = 0)$, the implementation replaces k/x by $k/\max(x, \sqrt{\varepsilon})$ where ε is the floating-point relative accuracy.

The laminar-turbulent transition at $\mathrm{Re}_x = 5.10^5$ is handled by the use of a smoothed Heaviside function with a transition of size 5.10^4, which corresponds to 10% of the threshold value.

CYLINDER IN CROSS-FLOW

Figure 4-37: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a cylinder in cross-flow.

This correlation is given by equation 7.54 in Ref. 21:

$$h = \frac{k}{D}\left(0.3 + \frac{0.62\,\mathrm{Re}_D^{1/2}\mathrm{Pr}^{1/3}}{\left(1 + \left(\frac{0.4}{\mathrm{Pr}}\right)^{2/3}\right)^{1/4}}\left(1 + \left(\frac{\mathrm{Re}_D}{282000}\right)^{5/8}\right)^{4/5}\right) \text{if } \mathrm{Re}_D\mathrm{Pr} \geq 0.2 \quad (4\text{-}187)$$

where $\mathrm{Pr} = \mu C_p/k$ and $\mathrm{Re}_D = \rho U D/\mu$. The cylinder diameter, D, and the exterior velocity, U, are correlation inputs. The material data are evaluated at $(T + T_{\mathrm{ext}})/2$.

SPHERE

Figure 4-38: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a sphere.

This correlation corresponds to equation 7.56 in Ref. 21:

$$h = \frac{k}{D}\left(2 + (0.4\,\mathrm{Re}_D^{1/2} + 0.06\,\mathrm{Re}_D^{2/3})\mathrm{Pr}^{0.4}\left(\frac{\mu}{\mu_s}\right)^{1/4}\right) \quad (4\text{-}188)$$

$$
\text{if} \begin{cases} 3.5 \leq \mathrm{Re}_D \leq 7.6 \cdot 10^4 \\ 0.71 \leq \mathrm{Pr} \leq 380 \\ 1 \leq \dfrac{\mu}{\mu(T)} \leq 3.2 \end{cases}
$$

where $\mathrm{Pr} = \mu C_p / k$ and $\mathrm{Re}_D = \rho U D / \mu$. The sphere diameter, D, and the exterior velocity, U, are correlation inputs. All material data are evaluated at T_{ext} except μ_s, which is evaluated at the wall temperature, T.

Heat Transfer Coefficients — Internal Forced Convection

ISOTHERMAL TUBE

Figure 4-39: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection in an isothermal circular tube.

This correlation corresponds to equations 8.55 and 8.61 in Ref. 21:

$$
h = \begin{cases} \dfrac{k}{D} 3.66 & \text{if } \mathrm{Re}_D \leq 2500 \\[2mm] \dfrac{k}{D} 0.027 \mathrm{Re}_D^{4/5} \mathrm{Pr}^{1/3} \left(\dfrac{\mu}{\mu_s} \right)^{0.14} & \text{if } \mathrm{Re}_D > 2500 \end{cases} \tag{4-189}
$$

where $\mathrm{Pr} = \mu C_p / k$, and $\mathrm{Re}_D = \rho U_{\text{ext}} D / \mu$. D (the tube diameter) and the velocity U are correlation inputs. All material data are evaluated at $(T + T_{\text{ext}})/2$ except μ_s, which is evaluated at the wall temperature, T. The laminar-turbulent transition at $\mathrm{Re}_D = 2500$ is handled by the use of a smoothed Heaviside function with a transition of size 250, which corresponds to 10% of the threshold value.

Using the Heat and Mass Transfer Analogy for the Evaluation of Moisture Transfer Coefficients

In the same way that the heat flux is described by the equation

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{ext} - T)$$

where h is a heat transfer coefficient and T_{ext} the temperature of the external fluid far from the boundary, the moisture flux is described by the equation

$$-\mathbf{n} \cdot \mathbf{g} = M_v h_m (c_{v,\,ext} - c_v) \tag{4-190}$$

where M_v is the molar mass of water vapor, h_m is a moisture transfer coefficient and $c_{v,ext}$ the vapor concentration of the external fluid far from the boundary.

In Ref. 21, the heat and mass boundary layer analogy is used to express the moisture transfer coefficient h_m as a function of the heat transfer coefficient h:

$$h_m = \frac{D\mathrm{Le}^n}{k}h = \frac{D}{k}\left(\frac{k}{\rho C_p D}\right)^n h \tag{4-191}$$

where Le is the Lewis number, D is the vapor diffusion coefficient in air, k is the thermal conductivity, ρ is the density, and C_p is the specific heat capacity of the external fluid evaluated at the same conditions as those specified in the heat transfer correlation used to compute h.

This relation is valid for laminar and turbulent external flows, and the value $n=1/3$ is suggested as a reasonable assumption for most applications in Ref. 21.

Note that the analogy applies only when the density variations due to temperature and vapor concentration have the same trend. In other words, if the hot air is heavier than cold air due to the moisture, the correlations are not suitable. Else, all the correlations described in Heat Transfer Coefficients — External Natural Convection, Heat Transfer Coefficients — Internal Natural Convection, Heat Transfer Coefficients — External Forced Convection, and Heat Transfer Coefficients — Internal Forced Convection may be used to evaluate a moisture transfer coefficient h_m to be used in Equation 4-190.

Nucleate Pool Boiling Correlation

Pool boiling is evaporation at a solid surface immersed in a bulk-stationary volume of liquid, where fluid motion is only due to natural convection, by opposition to flow boiling, for which forced convection is considered.

Different regimes of pool boiling may be distinguished, depending on the difference between the surface temperature and the saturation temperature of the boiling fluid:

- When the surface temperature is above the saturation temperature of the fluid by less than 10 K, heat transfer is moderate as it mainly occurs between the surface and the liquid phase of the fluid by convection, due to the formation of isolated bubbles at nucleation sites separating from the surface.

- For a temperature difference above 10 K, more nucleation sites become active. This is called the nucleate boiling regime, at which the heat transfer coefficient can reach much larger values. When temperature difference further increases, it causes interference and coalescence of the bubbles, creation of jets and columns, decreasing fluid motion and convective heat transfer. The critical heat flux value is reached when the effect of these interferences can't be compensated by an increase of the temperature difference. This happens for a temperature difference of about 30 K for water.

Rohsenow's Correlation for Pool Nucleate Boiling

The Rohsenow's correlation (Ref. 21) gives an estimation of the heat flux due to nucleate boiling. It reads

$$-\mathbf{n} \cdot \mathbf{q} = -\mu_l L_v \left(\frac{g(\rho_l - \rho_v)}{\sigma} \right)^{1/2} \left[\frac{C_{p,l}(T - T_{\text{sat}})}{\left(\frac{C_{p,l}\mu_l}{k_l} \right)^s C_{\text{sf}} L_v} \right]^3 \tag{4-192}$$

where:

- μ_l, ρ_l, k_l, and $C_{p,l}$ are the viscosity, density, thermal conductivity, and heat capacity of the liquid phase of the fluid, measured at T_{sat}

- ρ_v is the density of the saturated vapor of the fluid

- L_v is the latent heat of vaporization

- g is the gravitational acceleration

- σ is the surface tension of the liquid in contact with its own vapor

- T is the temperature of the solid surface

- T_{sat} is the saturation temperature of the fluid

- s is an empirical constant, equal to 1 for water, and set to 1.7 for other fluids in Ref. 21, Ref. 41, and Ref. 42

- C_{sf} is an empirical constant accounting for the combination of liquid and surface materials, and for the surface finishing. See Ref. 21, Ref. 41, and Ref. 42 for values of this constant for several combinations.

Note that the correlation is valid for values of gravitational acceleration from 1 to 10 m/s^2, and for clean surfaces. It is insensitive to the shape and orientation of the surface.

Critical Heat Flux for Nucleate Boiling

The correlation given in Equation 4-192 is valid up to a temperature difference at which the critical heat flux is obtained. The critical heat flux can be expressed in function of the latent heat, liquid and vapor densities, surface tension, and gravitational acceleration as follows (Ref. 21):

$$q_{0,\,max} = C_{max} L_v \rho_v^{1/2} [\sigma g (\rho_l - \rho_v)]^{1/4} \tag{4-193}$$

where C_{max} =0.149 for a large horizontal surface, and C_{max} =0.116 for a large horizontal cylinder.

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.

> 📄 See Convectively Enhanced Conductivity node.

Correlations giving Nu from various material properties for two configurations of rectangular enclosures are described below. These correlations are based on the Rayleigh number

$$\mathrm{Ra} = \frac{\rho^2 g |\alpha_p| C_p}{\mu k} \Delta T L^3$$

with α_p the coefficient of thermal expansion defined as

$$\alpha_p = -\frac{1}{\rho}\left(\frac{\partial \rho}{\partial T}\right)_p$$

where ρ is the fluid density, g the gravitational acceleration, C_p the heat capacity, μ the dynamic viscosity, k the thermal conductivity coefficient, ΔT the temperature difference through the cavity, and L the characteristic dimension of the cavity. The temperature difference through the cavity may be unknown *a priori*. In this case, ΔT can be defined using the **Automatic** option. The **Automatic** option defines a minimum and a maximum operator on the external boundaries of the domains where the correlation is used. As this option is designed for fluid domains where no heat source is defined, the difference between the maximum and the minimum of the temperature corresponds to the temperature difference through the cavity.

Note that only thermally induced variations of the density are accounted for in this definition. In moist air in particular, the natural convection due to vapor concentration is neglected.

Horizontal Cavity With Bottom Heating
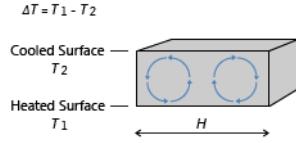
Figure 4-40: Schematic representation of geometry and parameters for the equivalent thermal conductivity correlation applied to an horizontal cavity heated from below.

The following correlation corresponds to equation 9.49 in Ref. 21. It is validated for $3 \cdot 10^5 \leq \mathrm{Ra}_H \leq 7 \cdot 10^9$:

$$\mathrm{Nu} \; = \; 0.069 \mathrm{Ra}_H^{1/3} \mathrm{Pr}^{0.074} \tag{4-194}$$

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
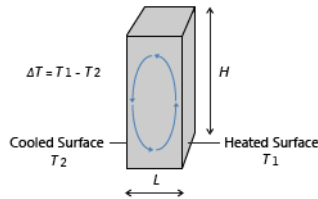
Figure 4-41: Schematic representation of geometry and parameters for the equivalent thermal conductivity correlation applied to a vertical cavity with sidewall heating.

The following correlations correspond to equations 9.50, 9.51, and 9.52 in Ref. 21:

- If $1 \leq H/L \leq 2$, $10^{-3} \leq \mathrm{Pr} \leq 10^5$, and $10^3 \leq \dfrac{\mathrm{Pr}\mathrm{Ra}_L}{0.2 + \mathrm{Pr}}$:

$$\text{Nu} = 0.18\left(\frac{\text{Pr}}{0.2 + \text{Pr}}\text{Ra}_L\right)^{0.29} \tag{4-195}$$

- If $2 \leq H/L \leq 10$, $\text{Pr} \leq 10^5$, and $10^3 \leq \text{Ra}_L \leq 10^{10}$:

$$\text{Nu} = 0.22\left(\frac{\text{Pr}}{0.2 + \text{Pr}}\text{Ra}_L\right)^{0.28}\left(\frac{H}{L}\right)^{-1/4} \tag{4-196}$$

- If $10 \leq H/L \leq 40$, $1 \leq \text{Pr} \leq 2 \cdot 10^4$, and $10^4 \leq \text{Ra}_L \leq 10^7$:

$$\text{Nu} = 0.42\text{Ra}_L^{1/4}\text{Pr}^{0.012}\left(\frac{H}{L}\right)^{-0.3} \tag{4-197}$$

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 $\text{Pr} = \mu C_p/k$. 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 the 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 4-5: COEFFICIENTS OF SURFACE TENSION

INTERFACE	A	B	REFERENCE
Acetone - Air	$-1.120 \cdot 10^{-4}$	$2.626 \cdot 10^{-2}$	Ref. 43
Acetic Acid - Air	$-0.994 \cdot 10^{-4}$	$2.958 \cdot 10^{-2}$	Ref. 43
Ethanol - Air	$-0.832 \cdot 10^{-4}$	$2.405 \cdot 10^{-2}$	Ref. 43
Toluene - Air	$-1.189 \cdot 10^{-4}$	$3.09 \cdot 10^{-2}$	Ref. 43
Diethyl Ether - Air	$-0.908 \cdot 10^{-4}$	$1.892 \cdot 10^{-2}$	Ref. 43
Glycerol - Air	$-0.885 \cdot 10^{-4}$	$6.517 \cdot 10^{-2}$	Ref. 43
Heptane - Nitrogen	$-0.980 \cdot 10^{-4}$	$2.21 \cdot 10^{-2}$	Ref. 44
Mercury - Mercury (Vapor)	$-2.049 \cdot 10^{-4}$	$49.06 \cdot 10^{-2}$	Ref. 44
Ethylene Glycol - Ethylene Glycol (Vapor)	$-0.890 \cdot 10^{-4}$	$5.021 \cdot 10^{-2}$	Ref. 44

More data can be found in Ref. 43 and Ref. 44.

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):

$$\mathbf{q}_{tot} = \rho\mathbf{u}E + \mathbf{q} + \mathbf{q}_r \tag{4-198}$$

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, $\mathbf{q}_r$. Hence, the total heat flux accounts for all three kinds of heat transfer described in The Physical Mechanisms Underlying 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:

$$E = H + \frac{1}{\rho}\mathbf{P}{:}\mathbf{F}$$

The total heat flux vector, $\mathbf{q}_{tot}$, is more suited to check the heat balance as described in the next section Heat and Energy Balance.

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. 40) — but it is in general rather difficult to derive. Potential energy is therefore often excluded and the total energy flux is approximated by

$$\mathbf{e}_{\mathrm{tot}} = \rho\mathbf{u}E_0 + \mathbf{q} + \mathbf{q}_{\mathrm{r}} - \sigma\mathbf{u} \qquad\qquad (4\text{-}199)$$

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 4-13. It expresses the idea that internal energy variations in time and net heat flux are balanced by external heat and work sources.

$$\frac{d}{dt}\int_{\Omega} \rho E dv + \int_{\partial\Omega_{\text{ext}}} \mathbf{q}_{\text{tot}} \cdot \mathbf{n} ds = Q_{\text{Int}} - W_{\text{str, Int}} \qquad (4\text{-}200)$$

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, $W_{\text{str, Int}}$, 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 4-200 is more visually represented by the diagram of Figure 4-42 below.
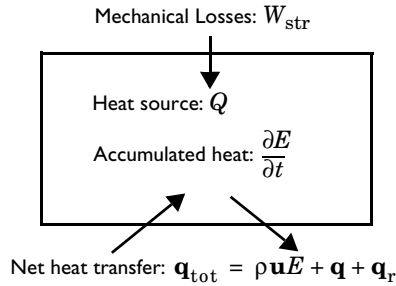
Figure 4-42: A heat balance diagram.

Several predefined variables are available in COMSOL Multiphysics to describe the heat rates involved in the system heat balance. See Global Variables for Heat and Energy Balance 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):

$$\frac{d}{dt}\int_{\Omega} \rho E_0 dv + \int_{\partial\Omega_{\text{ext}}} \mathbf{e}_{\text{tot}} \cdot \mathbf{n} ds = Q_{\text{Int}} + W_{\text{Int}} \qquad (4\text{-}201)$$

The variables in this formula are defined in Total Heat Flux and Energy Flux. Equation 4-201 is more visually represented by the diagram of Figure 4-43 below.
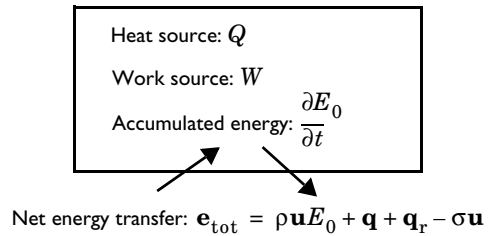
Figure 4-43: An energy balance diagram.

Several predefined variables are available in COMSOL Multiphysics to describe the energy rates involved in the system energy balance. See Global Variables for Heat and Energy Balance 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 in the spatial frame, and depending on the features, the user inputs may be defined in 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

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*:

$$\mathbf{F} = \begin{vmatrix} \dfrac{\partial x}{\partial X} & \dfrac{\partial x}{\partial Y} & \dfrac{\partial x}{\partial Z} \\[8pt] \dfrac{\partial y}{\partial X} & \dfrac{\partial y}{\partial Y} & \dfrac{\partial y}{\partial Z} \\[8pt] \dfrac{\partial z}{\partial X} & \dfrac{\partial z}{\partial Y} & \dfrac{\partial z}{\partial Z} \end{vmatrix} \tag{4-202}$$

It relates elementary distances $d\mathbf{x}$ and $d\mathbf{X}$ in the domain, expressed in material and spatial frames, according to:

$$d\mathbf{x} = \mathbf{F}d\mathbf{X} \tag{4-203}$$

The determinant of the deformation gradient, $\det(\mathbf{F})$, is the *volume ratio* field. In COMSOL Multiphysics, $\det(\mathbf{F})$ 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 $\mathbf{F}$.

	• 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: kg/m^3),
- Heat source, Q_0 (SI unit: W/m^3),
- Production/absorption coefficient, q_s (SI unit: $W/(m^3 \cdot K)$),
- Heat flux, q_0 (SI unit: W/m^2),
- Heat transfer coefficient, h (SI unit: $W/(m^2 \cdot K)$).

For all these variables, the conversion between the material and the spatial frame follows the relation:

$$A_{(x, y, z)} = \frac{A_{(X, Y, Z)}}{\det(\mathbf{F})}$$

This way, the integral of volumetric quantities over the domain, such as the mass density, is invariant between frames:

$$\int_{\Omega_0} \rho_{(X, Y, Z)} d\tau_0 = \int_{\Omega_0} \rho_{(x, y, z)} \det(\mathbf{F}) d\tau_0 = \int_{\Omega} \rho_{(x, y, z)} d\tau$$

In these equalities, Ω_0 and Ω denote the same domain but represented in the material frame or the 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 for heat flux and heat transfer coefficients:

$$\int_{\partial\Omega_0} h_{(X, Y, Z)} ds_0 = \int_{\partial\Omega_0} h_{(x, y, z)} \det(\mathbf{F}) ds_0 = \int_{\partial\Omega} h_{(x, y, z)} ds$$

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 the material and spatial frames, $\mathbf{u}_{(X, Y, Z)}$ and $\mathbf{u}_{(x, y, z)}$, is

$$\mathbf{u}_{(x, y, z)} = \mathbf{F}\mathbf{u}_{(X, Y, Z)}$$

This is directly deduced from the differential relation of Equation 4-203.

THERMAL CONDUCTIVITY

Thermal conductivity, k, is a tensor density. The relationship between the value on the spatial frame and the material frame is:

$$k_{(x, y, z)} = \frac{1}{\det(\mathbf{F})}\mathbf{F}k_{(X, Y, Z)}\mathbf{F}^{\mathrm{T}}$$

With this relation, and recalling that

$$\mathbf{n}_{(X, Y, Z)} = \mathbf{F}^{\mathrm{T}}\mathbf{n}_{(x, y, z)}$$

$$\nabla_{(X, Y, Z)}T = \mathbf{F}^{\mathrm{T}}\nabla_{(x, y, z)}T$$

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.

When the material frame differs from the spatial frame, the domain features representing solid domains give the option to specify the thermal conductivity either on the spatial or the material frame. See Heat Conduction, Solid for details about this option.

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 $\mathbf{F}_{\mathrm{xdim}}$ is equal to the deformation gradient tensor $\mathbf{F}$ defined in Equation 4-202.

- When the layer is conductive, the deformation gradient tensor $\mathbf{F_t}$ is defined using tangential derivatives as follows:

$$\mathbf{F_t} = \begin{bmatrix} xT_X & xT_Y & xT_Z \\ yT_X & yT_Y & yT_Z \\ zT_X & zT_Y & zT_Z \end{bmatrix}$$

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 $\mathbf{F}_{xdim}$ is defined as follows:

$$\mathbf{F}_{xdim} = \begin{bmatrix} xT_X + n_x n_X & xT_Y + n_x n_Y & xT_Z + n_x n_Z \\ yT_X + n_y n_X & yT_Y + n_y n_Y & yT_Z + n_y n_Z \\ zT_X + n_z n_X & zT_Y + n_z n_Y & zT_Z + n_z n_Z \end{bmatrix}$$

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 derivatives of physical quantities such as temperature or internal energy in heat transfer. The variations of such derivatives during an elementary time step are studied for the 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 in the case 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. Harry H. Pennes, *Analysis of Tissue and Arterial Blood Temperatures in the Resting Human Forearm*, Journal of Applied Physiology, Vol. 1 no. 2, pp. 93-122, 1 August 1948.

7. 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.

8. 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.

9. 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.

10. 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 (https://pubmed.ncbi.nlm.nih.gov/12953912/).

11. 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.
https://itis.swiss/virtual-population/tissue-properties/overview/

12. 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.

13. D.A. Nield and A. Bejan, *Convection in Porous Media*, in Convection Heat Transfer, Fourth Edition, John Wiley & Sons, Inc., Hoboken, NJ, USA, 2013.

14. B. Sundén and J. Yuan, *Evaluation of Models of the Effective Thermal Conductivity of Porous Materials Relevant to Fuel Cell Electrodes*, Int. J. Comp. Meth. and Exp. Meas., Vol 1 (4), 2013.

15. N. Wakao, S. Kaguei, and T. Funazkri, *Effect of fluid dispersion coefficients on particle-to-fluid heat transfer coefficients in packed beds*, Chem. Engng Sci. 34, pp. 325-336, 1979.

16. EN 15026, *Hygrothermal performance of building components and building elements - Assessment of moisture transfer by numerical simulation*, CEN, 2007.

17. J. Bear and Y. Bachmat, *Introduction to Modeling of Transport Phenomena in Porous Media*, Kluwer Academic Publisher, 1990.

18. H. Lee, *Thermal Design, Heat Sinks, Thermoelectrics, Heat Pipes, Compact Heat Exchangers, and Solar Cells*, John Wiley & Sons, 2006.

19. G. Fraisse, J. Ramousse, D. Sgorlon, C. Goupil, *Comparison of different modeling approaches for thermoelectric elements*, Energy Conversion and Management, Vol. 65, pp. 351-356, 2013.

20. R. Siegel and J. Howell, *Thermal Radiation Heat Transfer*, 4th ed., Taylor & Francis, New York, 2002.

21. 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.

22. https://www.esrl.noaa.gov/gmd/grad/solcalc/

23. M.F. Modest, *Radiative Heat Transfer*, 2nd ed., Academic Press, San Diego, California, 2003.

24. W.A. Fiveland, "The Selection of Discrete Ordinate Quadrature Sets for Anisotropic Scattering," *Fundamentals of Radiation Transfer*, HTD, vol. 160, ASME, 1991.

25. A. Bejan et al., *Heat Transfer Handbook*, John Wiley & Sons, 2003.

26. F. Charron, *Partage de la chaleur entre deux corps frottants*, Publication Scientifique et Technique du Ministère de l'Air, no. 182, 1943. (In French)

27. D.C. Wilcox, *Turbulence Modeling for CFD*, 2nd ed., DCW Industries, 1998.

28. 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.

29. W.M. Kays, "Turbulent Prandtl Number — Where Are We?", *ASME J. Heat Transfer*, vol. 116, pp. 284–295, 1994.

30. 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.

31. 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.

32. 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.

33. 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.

34. 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.

35. 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.

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

37. 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.

38. 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.

39. W. Wagner and H-J Kretzschmar, *International Steam Tables*, 2nd ed., Springer, 2008.

40. M. Kaviany, *Principles of Convective Heat Transfer*, 2nd ed., Springer, 2001.

41. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1993.

42. I. Pioro, *Experimental Evaluation of Constants for the Rohsenow Pool Boiling Correlation*, International Journal of Heat and Mass Transfer, Vol 42 (11), pp. 2003-2013, 1999.

43. 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

44. J.J. Jasper, "The Surface Tension of Pure Liquid Compounds", *J. Phys. Chem. Ref. Data*, vol. 1, pp. 841–1010, 1972.

45. G.K. Batchelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.

46. C. P. Thurgood, A. Pollard and H. A. Becker, "The T_N quadrature set for the discrete ordinates method," *J. Heat Transfer*, vol. 117, pp. 1068–1070, 1995.

47. M. A. Badri, P. Jolivet, B. Rousseau, S. Le Corre, H. Digonnet and Y. Favennec, "Vectorial finite elements for solving the radiative transfer equation," *Journal of Quantitative Spectroscopy and Radiative Transfer*, vol. 212, pp. 59–74, 2018.

5

The Heat Transfer Module Interfaces

The Heat Transfer Module includes several 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 in Solids
		Heat Transfer in Fluids
		Heat Transfer in Solids and Fluids
		Bioheat Transfer
		Heat Transfer in Porous Media
		Local Thermal Nonequilibrium
		Heat Transfer in Packed Beds
		Heat Transfer in Moist Porous Media
		Heat Transfer in Building Materials
		Heat Transfer in Moist Air
Heat Transfer in Shells	htlsh	Heat Transfer in Shells
		Heat Transfer in Films
		Heat Transfer in Fractures

TABLE 5-1: THE HEAT TRANSFER MODULE INTERFACES

INTERFACE	TAG	VERSIONS
Lumped Thermal System	lts	Lumped Thermal System
Surface-to-Surface Radiation	rad	Surface-to-Surface Radiation
Radiation in Participating Media	rpm	Radiation in Participating Media
Radiation in Absorbing-Scattering Media	rasm	Radiation in Absorbing-Scattering Media
Radiative Beam in Absorbing Media	rbam	Radiative Beam in Absorbing Media
Moisture Transport	mt	Moisture Transport in Air
		Moisture Transport in Porous Media
		Moisture Transport in Building Materials

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 in Solids and Fluids Interface
- The Heat Transfer in Porous Media Interface
- The Local Thermal Nonequilibrium Interface
- The Heat Transfer in Packed Beds Interface
- The Heat Transfer in Moist Porous Media Interface
- The Heat Transfer in Building Materials Interface
- The Heat Transfer in Moist Air Interface
- The Bioheat Transfer Interface
- The Heat Transfer in Shells Interface
- The Heat Transfer in Films Interface
- The Heat Transfer in Fractures Interface
- The Lumped Thermal System Interface

- The Surface-to-Surface Radiation Interface
- The Radiation in Participating Media Interface
- The Radiation in Absorbing-Scattering Media Interface
- The Radiative Beam in Absorbing Media Interface
- The Moisture Transport in Air Interface
- The Moisture Transport in Porous Media Interface
- The Moisture Transport in Building Materials 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 Shells (with several versions)
- Surface-to-Surface Radiation
- Radiation in Participating Media
- Radiation in Absorbing-Scattering Media
- Radiation with Radiative Beam in Absorbing Media
- Lumped Thermal System
- Moisture Transport (with several versions)

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 Shells apply in domains, with features available at each geometric level (volumes, surfaces, edges, and points).

The Heat Transfer in Shells interface apply on boundaries, with features available on surfaces, edges, and points.

The Lumped Thermal System interface is a global interface, with features not attached to any geometric entity.

Study Types

Stationary, time-dependent, and frequency domain 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 or a frequency domain study.

📑	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 in Solids and Fluids Interface
- The Bioheat Transfer Interface
- The Heat Transfer in Porous Media Interface
- The Local Thermal Nonequilibrium Interface
- The Heat Transfer in Moist Porous Media Interface
- The Heat Transfer in Building Materials Interface
- The Heat Transfer in Moist Air 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.

- If **Heat Transfer in Solids in Fluids** (⟨≋⟩) is selected, a **Heat Transfer in Solids and Fluids (ht)** interface is added with 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, a check box is available under the **Physical Model** section to allow for isothermal domain modeling.

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 Shells Physics Interface

The versions of the Heat Transfer in Shells physics interface (htlsh) are:

- The Heat Transfer in Shells Interface
- The Heat Transfer in 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 Shells interface is added. Depending on the version of the physics interface selected, the default nodes vary:

- If **Heat Transfer in Shells** (⟨▯⟩) is selected, a **Heat Transfer in Shells (htlsh)** interface is added with a default **Solid** model.

- If **Heat Transfer in Films** (⟨▱⟩) is selected, a **Heat Transfer in Films (htlsh)** interface is added with a default **Fluid** model.

- If **Heat Transfer in Fractures** (⟨▨⟩) is selected, a **Heat Transfer in Fractures (hltsh)** interface is added with a default **Porous Medium** model.

Benefits of the Different Heat Transfer in Shells Interfaces

The benefit of the different versions of the Heat Transfer in Shells interface, with `htlsh` 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 **Solid** node from the **Physics** toolbar — the functionality is always available.

TABLE 5-2: THE HEAT TRANSFER IN SHELLS (HTLSH) INTERFACE OPTIONS

ICON	NAME	DEFAULT PHYSICAL MODEL
	Heat Transfer in Shells	No check box is selected.
	Heat Transfer in Films	No check box is selected.
	Heat Transfer in Fractures	The Heat transfer in porous media check box is selected.

Layer and Interface Selection Tools

Almost all physics nodes that you add in the Heat Transfer in Shells interface have either a section named **Shell Properties**, or a section named **Interface Selection**. Selections there interact with the standard selections of geometrical objects (boundaries and edges) in order to provide a complete specification of where a material property or boundary condition is to be applied within a layered material.

The physics features can be of two, fundamentally different, types: those who act on the layers themselves, and those who act on the interfaces between layers. The settings for these two types of features are slightly different.

LAYER SELECTIONS

When a physics feature acts on layers, the interpretations of layer selections are:

- A boundary selection in the geometry corresponds to a domain contribution in the selected layers.
- An edge selection in the geometry corresponds to boundary contributions on the faces of the layers exposed to that edge.

This is illustrated by Figure 5-1 and Figure 5-2, where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.
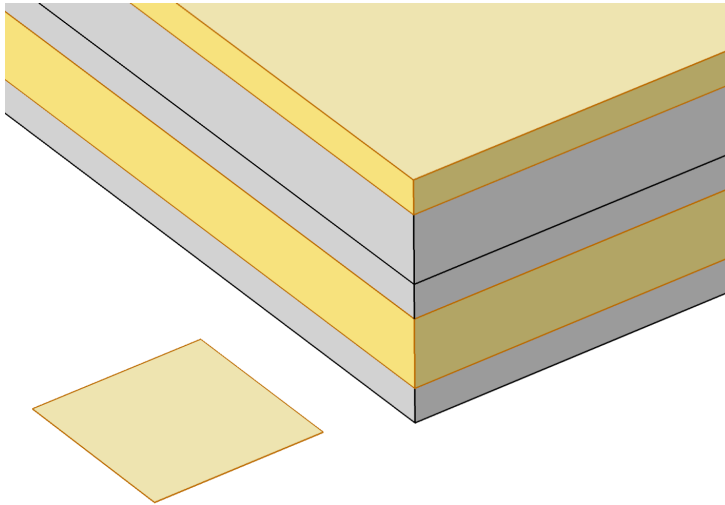
Figure 5-1: Boundary selection with layers 2 and 5 selected.
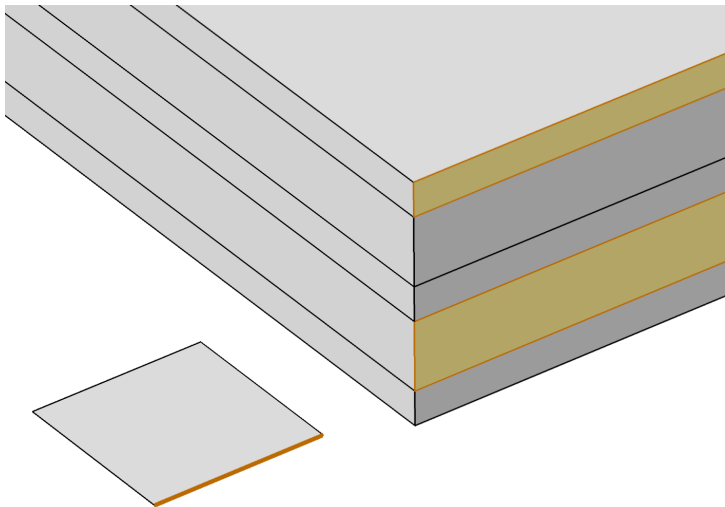
Figure 5-2: Edge selection with layers 2 and 5 selected.

In the **Shell Properties** section, it is possible to limit the contributions to individually selected layers of the selections made in the **Boundary Selection** section. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

INTERFACE SELECTIONS

When a physics feature acts on interfaces, the interpretation of interface selections are:

- A boundary selection in the geometry corresponds to boundary contributions on the selected interfaces.

- An edge selection in the geometry corresponds to edge contributions on the interface edges parallel to the selected edge.

This is illustrated by Figure 5-3 and Figure 5-4, where the patch at the lower left is the actual shell, and the 3D sketch shows an expanded view of what it would represent in the physical world.
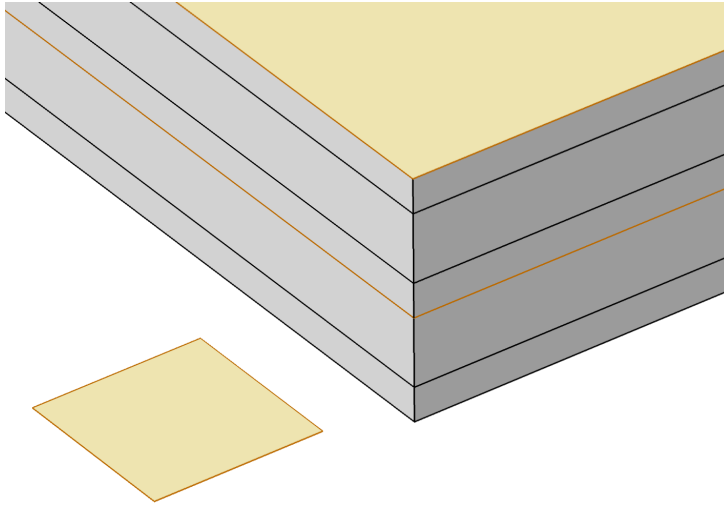
Figure 5-3: Boundary selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.
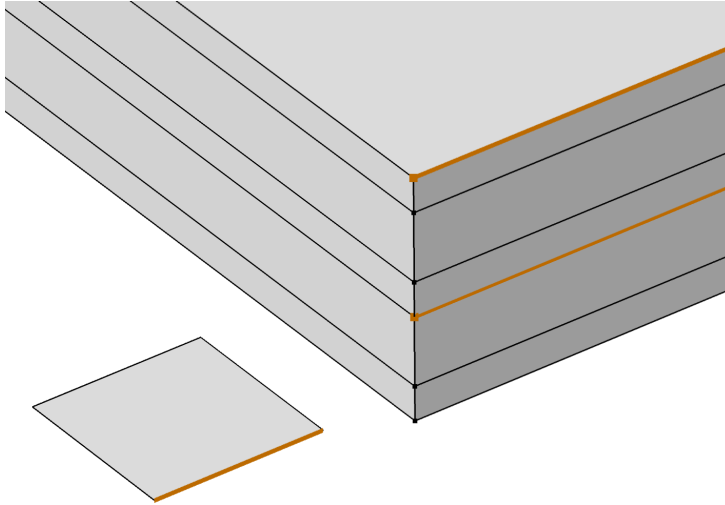
Figure 5-4: Edge selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

In the **Interface Selection** section, it is possible to select the interfaces between layers where the contribution should be added, including the top and bottom interfaces (see

[Figure 5-5](#) for the interfaces description). For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of predefined interfaces subsets.
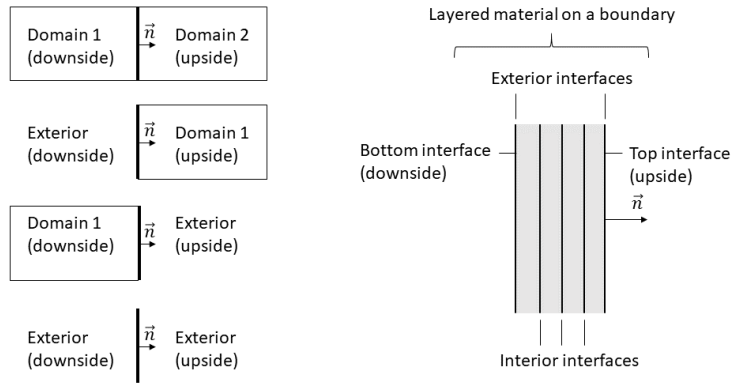
Figure 5-5: Definition of upside and downside settings for a boundary for different configurations, and corresponding interfaces for a layered material.

> 🔍 See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

Using the Extra Dimension Coordinates

It is possible to write expressions that are functions of the coordinates in the thickness direction of the shell. Expressions based on the usual coordinates, like x, y, and z, are evaluated on the reference surface (the meshed boundaries). In addition to this, the locations in the through-thickness direction can be accessed by making explicit or implicit use of the coordinates in the extra dimension.

The extra dimension coordinate has a name like x_llmat1_xdim. The middle part of the coordinate name is derived from the tag of the layered material definition where it is created; in this example a **Layered Material Link**.

Versions of the Moisture Transport Physics Interface

The versions of the Moisture Transport physics interface (mt) are:

• The Moisture Transport in Air Interface

- The Moisture Transport in Porous Media Interface
- The Moisture Transport in Building Materials Interface

After selecting a version, default nodes are added under the main node, which then defines which version of the Moisture Transport interface is added. Depending on the version of the physics interface selected, the default nodes vary:

- If **Moisture Transport in Air** ( ) is selected, a **Moisture Transport in Air (mt)** interface is added with a default **Moist Air** model.
- If **Moisture Transport in Building Materials** ( ) is selected, a **Moisture Transport in Building Materials (mt)** interface is added with a default **Building Material** model.
- If **Moisture Transport in Porous Media** ( ) is selected, a **Moisture Transport in Porous Media (mt)** interface is added with a default **Hygroscopic Porous Medium** model.

Benefits of the Different Moisture Transport Interfaces

The benefit of the different versions of the Moisture Transport interface, with mt as the common default name (see Moisture Transport Variables), is that it is easy to add the default settings when selecting the physics interface. At any time, add a **Building Material**, an **Hygroscopic Porous Medium**, or a **Moist Air** node from the **Physics** toolbar — the functionality is always available.

Additional Physics Options

Then, additional physics options are provided with the other interfaces and multiphysics interfaces:

TABLE 5-3: ADDITIONAL HEAT TRANSFER PHYSICS OPTIONS

ICON	NAME	ID
	Laminar Flow (under the **Conjugate Heat Transfer** branch)	—
	Turbulent Flow, Algebraic yPlus, Turbulent Flow, L-VEL, Turbulent Flow, k-ε, Turbulent Flow, Low Re k-ε (under the **Conjugate Heat Transfer** branch)	—
	Surface-to-Surface Radiation (under the **Radiation** branch)	rad
	Radiation in Participating Media (under the **Radiation** branch)	rpm
	Radiation in Absorbing-Scattering Media (under the **Radiation** branch)	rasm
	Radiative Beam in Absorbing Media (under the **Radiation** branch)	rbam
	Joule Heating (under the **Electromagnetic Heating** branch)	—

TABLE 5-3: ADDITIONAL HEAT TRANSFER PHYSICS OPTIONS

ICON	NAME	ID
	Heat and Moisture	—
	Thermoelectric Effect	—

More turbulent flow interfaces are available under the **Conjugate Heat Transfer** branch with the CFD Module:

- Turbulent Flow, Realizable k-ε
- Turbulent Flow, k-ω
- Turbulent Flow, SST
- Turbulent Flow, Spalart-Allmaras
- Turbulent Flow, v2-f

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 a license for the AC/DC Module, Wave Optics Module, and RF Module.

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 $2\sqrt{\pi A_c}$, respectively.

<table>
<tr><td>📋</td><td>When the Heat Transfer interface is coupled to an interface modeling radiation (Surface-to-Surface Radiation, Radiation in Participating Media, Radiation in Absorbing-Scattering Media), through a multiphysics coupling (**Heat Transfer with Surface-to-Surface Radiation**, **Heat Transfer with Radiation in Participating Media**, or **Heat Transfer with Radiation in Absorbing-Scattering Media**), these inputs are automatically defined from the multiphysics coupling. These variables are set to unit length of the component unit system. This corresponds to the assumption that the geometry is infinite in the out-of-plane direction and that the equations are defined per unit length. This assumption is required as it corresponds to the view factor computation in these dimensions.</td></tr>
</table>

Set the **Reference temperature** T_{ref}. It is used for the evaluation of the reference density ρ_{ref} when the Heat Transfer interface is coupled through **Nonisothermal Flow** multiphysics coupling to a Fluid Flow interface with **Compressibility** set to **Incompressible** and **Specify reference temperature** set to **From heat transfer interface**. When **Specify reference temperature** is set to **From fluid flow interface** in **Nonisothermal Flow** coupling node, this input is synchronized with the corresponding input in the Single Phase Flow interface. It is also used in the definition of the reference enthalpy H_{ref} which is set to 0 J/kg at p_{ref} (1 atm) and T_{ref}. The corresponding interface variable is `ht.Tref`.

Select the **Isothermal domain** check box to activate the support for isothermal domain modeling. See Isothermal Domain Interface and Isothermal Domain.

<table>
<tr><td>🔍</td><td>For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/</td></tr>
</table>

CONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button (👁) and selecting **Stabilization** in the **Show More Options** dialog box.

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. See Heat Transfer Consistent and Inconsistent Stabilization Methods for details.

DAMAGE INTEGRAL ANALYSIS DISCRETIZATION

When the **Thermal Damage** subnode is added under **Biological Tissue**, select the type of the **Shape function for damaged tissue indicators**. The default is **Discontinuous Lagrange**. The order is set in the Discretization section.

DISCRETIZATION

To display all settings available in this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. You can choose the type and order of the shape functions used for the variables solved by the Heat Transfer interfaces.

Temperature

For the temperature, you can choose not only the order of the discretization, but also the type of shape functions: *Lagrange* or *serendipity*. For highly distorted elements, Lagrange shape functions provide better accuracy than serendipity shape functions of the same order. The serendipity shape functions will however give significant reductions of the model size for a given mesh containing hexahedral, prism, or quadrilateral elements. At first order, Lagrange and serendipity shape functions are the same.

The default shape functions used for the temperature are **Quadratic Lagrange** for the modeling of heat transfer without convection (solids, biological tissue, building materials), and **Linear** for the modeling of heat transfer with convection (fluids, porous media, moist air). See each interface description for details.

Damaged Tissue Indicator

When the **Thermal Damage** subnode is added under **Biological Tissue**, you can choose the discretization level of the **Damage tissue indicator** shape function: **Constant** (the default), **Linear**, **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

Irreversible Transformation Indicator

When the **Irreversible Transformation** subfeature is added under a **Solid** node, you can choose the discretization level of the **Irreversible transformation indicator** shape function: **Constant** (the default), **Linear**, **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

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.

Feature Nodes for the Heat Transfer Interface

This section details the nodes available within the Heat Transfer interface:

- Domain Nodes for the Heat Transfer Interface
- Boundary Nodes for the Heat Transfer Interface
- Edge Nodes for the Heat Transfer Interface
- Point Nodes for the Heat Transfer Interface

	Some nodes are only available with some COMSOL products.
🔍	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

The Heat Transfer interface has the following domain nodes:

- Bioheat
- Biological Tissue
- Building Material
- Convectively Enhanced Conductivity
- Cross Section
- Fluid
- Fluid (Porous Medium)
- Geothermal Heating
- Heat Source
- Immobile Fluids (Porous Medium)
- Initial Values
- Irreversible Transformation
- Liquid Water (Moist Porous Medium)
- Moist Air (Heat Transfer Interface)
- Moist Air (Moist Porous Medium)
- Moist Porous Medium
- Optically Thick Participating Medium

- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Pellets (Porous Medium)
- Pellet-Fluid Interface (Porous Medium)
- Phase Change Material
- Porous Matrix (Porous Medium, Moist Porous Medium)
- Porous Medium
- Pressure Work
- Shape Memory Alloy
- Solid
- Thermal Damage
- Thermal Dispersion
- Thermoelastic Damping
- Thickness
- 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.

The Heat Transfer interface has the following boundary nodes:

- Boundary Heat Source
- Continuity
- Deposited Beam Power
- External Temperature (Thin Layer, Thin Film, Fracture)
- Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
- Heat Flux
- Heat Source (Thin Layer, Thin Film, Fracture)
- Inflow
- Line Heat Source on Axis
- Local Thermal Nonequilibrium Boundary
- Lumped System Connector
- Open Boundary

- Outflow
- Periodic Condition (Heat Transfer Interface)
- Phase Change Interface
- Surface-to-Ambient Radiation (Heat Transfer Interface)
- Symmetry (Heat Transfer Interface)
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)

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.

The Heat Transfer interface has the following edge nodes (3D components only):

- Heat Flux (Thin Layer, Thin Film, Fracture)
- Line Heat Source
- Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thin Rod

The Heat Transfer interface has the following point nodes:

- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis

- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Rod)

Settings for the Heat Transfer in 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 htlsh.

BOUNDARY SELECTION

By default, all boundaries are available for the application of the Heat Transfer in Shells interface. Select the **Restrict to layered boundaries** check box to make the interface applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or

Layered Material Stack) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

SHELL PROPERTIES

Two options are available for the **Shell type**:

- When the **Layered shell** option is selected, the Extra Dimension tool is used to solve the equations through the thickness of a layered material. It is possible to consider several layers with different thermal properties varying through the thickness, by using the **General** option for **Layer type** in the Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface), Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface), or Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface) nodes.

- When the **Nonlayered shell** option is selected, only the thermal properties need to be specified within the material. This option should be used for thermally thin layers, for which no through-thickness temperature variation is expected in the layered material. This lumped approach is available by using the **Thermally thin approximation** option for **Layer type** in the Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface), Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface), or Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface) nodes.

Layered Shell

By default, the **Shell type** is **Layered shell**, and the thickness of the layered material should be set as follows, depending on the type of material:

- In a **Material** node, the **Layer thickness** can be in the table found under the **Material Contents** section of the material **Settings** window. This automatically adds a **Shell** subnode under the **Material** node, transforming it as a layered material.

- When the layered material is a **Single Layer Material**, the **Thickness** is set in the **Layer Definition** section of the **Shell Property Group** window.

- For a general **Layered Material**, added through a **Layered Material Link** or a **Layered Material Stack**, the **Thickness** is set in the **Layer Definition** section of the **Settings** window. Several layers may be defined in the table, and the **Thickness** should be defined for each of them. The total thickness of the layered material is the sum of all the layers thicknesses.

Note that the **Layered shell** option should be used whenever a layered material is applied on the boundaries, because the thickness is part of the material settings.

In the **Solid**, **Fluid**, and **Porous Medium** nodes, the same layered material is used, and this choice is not editable. And both the **Thermally thin approximation** and **General** options are available as **Layer type** in these nodes.

Clear the **Use all layers** check box to apply the Heat Transfer in Shells interface on some layers only. Select a **Layered material** from the list (the interface is then applicable only on the boundaries where this latter material is defined), and clear the check boxes corresponding to layers where the interface should not be applied in the **Selection** table.

Nonlayered Shell

If the **Restrict to layered boundaries** check box is not selected in the **Boundary Selection** section, a nonlayered material may be defined on the selected boundaries, and the **Thickness** L_{th} can be set as a user defined value or expression. This value overrides the values set in the material nodes.

In the **Solid**, **Fluid**, and **Porous Medium** nodes, the **Thickness** is set by default to **From physics interface**, and is editable only in a manually added node. Only the **Thermally thin approximation** option is available as **Layer type** in these nodes.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer cross section preview** and **Layer 3D preview** buttons.

ℚ	• The layer thickness variable, `htlsh.ds`, used in the weak equations, is the product of the thickness set in the **Shell Properties** section, `htlsh.lth`, and the scale factor `htlsh.lsc`, which is equal to 1 by default, and can be overridden by a user defined value in a single layer material.
	• The layer thickness variable, `htlsh.ds`, is defined in all dimensions, and is unrelated to the **Out-of-plane thickness** of the layered material, `htlsh.d`, which is only available in 2D, and might be edited in the Out-of-Plane Thickness section.

ℚ	• See Layer and Interface Selection Tools.
	• You can provide material parameters with a through-thickness variation by explicitly or implicitly using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

The desired selection for the physics interface may correspond to boundaries with different layered materials. This means that a single physics interface can accommodate several different stacking sequences.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

OUT-OF-PLANE THICKNESS

For 2D components, the cross-section of the layered material is modeled, and its **Out-of-plane thickness**, d_z (SI unit: m), should be defined (see Equation 4-72). The default is 1 m.

When the Heat Transfer in Shells interface is coupled to a **Surface-to-Surface Radiation** interface through a **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling, these inputs are automatically defined from the multiphysics coupling. These variables are set to unit length of the component unit system. This corresponds to the assumption that the geometry is infinite in the out-of-plane direction and that the equations are defined per unit length. This assumption is required as it corresponds to the view factor computation in these dimensions.

PHYSICAL MODEL

Set the **Reference temperature** T_{ref}. It is used in the definition of the reference enthalpy H_{ref} which is set to 0 J/kg at p_{ref} (1 atm) and T_{ref}. The corresponding interface variable is htlsh.Tref.

This input is not overriden by the model input found in features like Solid, Porous Medium, Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface), and Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface). It sets the ht.Tref variable.

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

DISCRETIZATION

To display all settings available in this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. You can choose the type and order of the shape functions used for the variables solved by the Heat Transfer in Shells interfaces.

Temperature

For the temperature, you can choose not only the order of the discretization, but also the type of shape functions: *Lagrange* or *serendipity*. For highly distorted elements, Lagrange shape functions provide better accuracy than serendipity shape functions of the same order. The serendipity shape functions will however give significant reductions of the model size for a given mesh containing hexahedral, prism, or quadrilateral elements.

The shape functions used for the temperature are **Quadratic Lagrange** for the modeling of heat transfer in shells, and **Linear** for the modeling of heat transfer in films and heat transfer in fractures.

DEPENDENT VARIABLES

The Heat Transfer in 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.

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 $2\sqrt{\pi A_c}$, respectively.

Set the **Reference vapor concentration** which is used when the moisture transport interface is coupled to a fluid flow interface with the **Incompressible flow** option selected in the **Compressibility** list. In this case, the fluid density is evaluated at the **Reference pressure level** (defined in the **Physical Model** section of the fluid flow interface), at the reference temperature defined in the moisture transport interface (293.15 K), and at the **Reference vapor concentration** specified in this section.

Choose the **Mixture type for moist air**:

- By default, a **Diluted species** formulation is used in the Moisture Transport in Air interface and in the Moisture Transport in Building Materials interface. This formulation is well-adapted for small vapor concentration conditions. In this case, the density of moist air is supposed not to change with variations of moisture content, and the moisture content variation is expressed through the transport of vapor concentration c_v

- For larger vapor concentration conditions, the variation of moist air density are modeled, and the **Concentrated species** formulation should be used. In this case, the moisture content variation is expressed through the transport of vapor mass fraction ω_v.

Note that in hygroscopic porous media domains, the **Concentrated species** formulation is always used, and this setting is not used. In building materials domains, this setting has no effect either.

Finally, select the **Account for gravity in liquid velocity** check box to enable the Gravity feature.

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 • Selection Information

ADVANCED SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**. These settings only apply to the domains in which either a **Moist Air** feature or an **Hygroscopic Porous Medium** feature is active. Select a **Convective term** — **Non-conservative form** (the default) or **Conservative form**. The latter should be used for compressible flow.

DISCRETIZATION

To display all settings available in this section, click the **Show More Options** button (👁) and select **Advanced Physics Options** in the **Show More Options** dialog box. The shape

functions used for the relative humidity are by default **Quadratic Lagrange** in building materials and porous media, and **Linear** in moist air.

| ⊕ | Theory for Moisture Transport |

Feature Nodes for the Moisture Transport Interface

This section details the nodes available with the Moisture Transport interface.

DOMAIN NODES FOR THE MOISTURE TRANSPORT INTERFACE

The Moisture Transport interface has the following domain nodes:

- Building Material
- Hygroscopic Porous Medium
- Initial Values
- Liquid Water (Hygroscopic Porous Medium)
- Moist Air (Hygroscopic Porous Medium)

- Moist Air (Moisture Transport Interface)
- Moisture Source
- Porous Matrix (Hygroscopic Porous Medium)
- Turbulent Mixing

BOUNDARY NODES FOR THE MOISTURE TRANSPORT INTERFACE

The Moisture Transport interface has the following boundary nodes:

- Continuity
- Inflow
- Insulation
- Moist Surface
- Moisture Content
- Moisture Flux

- Open Boundary
- Outflow
- Symmetry
- Thin Moisture Barrier
- Wet Surface

The Heat Transfer in Solids Interface

The **Heat Transfer in Solids (ht)** 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 in Solids** to select physics features from the context menu.

By default, the shape functions used for the temperature in solids are **Quadratic Lagrange**.

📋	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for a description of all the settings and functionality.

🔍	• Handling Frames in Heat Transfer • Theory for Heat Transfer in Solids

The Heat Transfer in Fluids Interface

The **Heat Transfer in Fluids (ht)** interface ($|\approx$) 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.

By default, the shape functions used for the temperature in fluids are **Linear**.

🗐	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

🔍	• Handling Frames in Heat Transfer • Theory for Heat Transfer in Fluids

The Heat Transfer in Solids and Fluids Interface

The **Heat Transfer in Solids and Fluids** (⟨≋⟩) 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 are the same as for The Heat Transfer in Fluids Interface. This means that a first order discretization of the temperature is set by default.

> 🗐 See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

The Heat Transfer in Porous Media Interface

The **Heat Transfer in Porous Media (ht)** interface (![icon]), found in the **Porous Media** physics area under the **Heat Transfer** branch (![icon]) when adding a physics interface, is used to model heat transfer by conduction and convection in porous media. A **Porous Medium** model is active by default on all domains, with **Porous medium type** set to **Local thermal equilibrium**. All functionality for including other domain types, such as a solid domain, is also available, and surface-to-ambient radiation may be considered.

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 Nonequilibrium 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.

By default, the shape functions used for the temperature in porous media are **Linear**.

![icon]	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

![icon]	Theory for Heat Transfer in Porous Media

![icon]	*Free Convection in a Porous Medium*: Application Library path **Subsurface_Flow_Module/Heat_Transfer/convection_porous_medium**

The Local Thermal Nonequilibrium Interface

The **Local Thermal Nonequilibrium** interface ( ), found in the **Porous Media** physics area under the **Heat Transfer** branch ( ) when adding a physics interface, is used to model heat transfer by conduction and convection, in porous media for which the solid and fluid phase temperatures are not in equilibrium. A **Porous Medium** model is active by default on all domains, with **Porous medium type** set to **Local thermal nonequilibrium**. All functionality for including other domain types, such as a solid domain, is also available, and surface-to-ambient radiation may be considered.

The temperature equations in the solid and fluid phases of the porous medium are solved and coupled through a transfer term proportional to the temperature difference between both phases. If this difference can be neglected, use the The Heat Transfer in Porous Media 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) overridden by **Local Thermal Nonequilibrium Boundary** (showing all the boundaries adjacent to domains where two temperatures are solved for solid and fluid phases), and **Initial Values**.

Specific subnodes are also present by default under the **Fluid** and **Porous Matrix** subnodes of the **Porous Medium** node:

• **Initial Values**, which allows to set a specific initial temperature for each phase

• **Thermal Insulation**, which sets a no flux condition for each phase

• **Continuity** (under **Fluid** subnode only), which ensures the continuity of the temperature between the fluid phase and the other domains.

Other subnodes implementing boundary conditions specific to the fluid and solid phases may be added, to model flow conditions, heat sources, fluxes, and phase change. This can be done by right-clicking **Fluid** or **Porous Matrix** to select physics features from the context menu.

Heat Sources

In the heat source features available under the **Fluid** and **Porous Matrix** subnodes, 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 under the **Fluid** and **Porous Matrix** subnodes, 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.

By default, the shape functions used for the temperature in porous media are **Linear**.

🗒	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

🔍	Theory for Heat Transfer in Porous Media

The Heat Transfer in Packed Beds Interface

The **Heat Transfer in Packed Beds** interface (![icon]), found in the **Porous Media** physics area under the **Heat Transfer** branch (![icon]) when adding a physics interface, is used to model heat transfer by conduction and convection, in porous media where the local thermal equilibrium is not assumed between the solid pellets of a packed bed and the fluid phase, and where the radial variation of the temperature due to conduction inside the pellets is accounted for. A **Porous Medium** model is active by default on all domains, with **Porous medium type** set to **Packed bed**. All functionality for including other domain types, such as a solid domain, is also available, and surface-to-ambient radiation may be considered, assuming that radiation effects are handled on the exterior surface of the pellets only.

The microscale heat equation in the pellets and the macroscale heat equation in the fluid are solved and coupled either through a transfer term proportional to the temperature difference between both phases, or a continuity condition on the fluid and pellets temperatures.

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) overridden by **Local Thermal Nonequilibrium Boundary** (showing all the boundaries adjacent to domains where two temperatures are solved for solid and fluid phases), and **Initial Values**.

Specific subnodes are also present by default under the **Fluid** and **Pellets** subnodes of the **Porous Medium** node:

- **Initial Values**, which allows to set a specific initial temperature for each phase
- **Thermal Insulation** (under **Fluid** subnode only), which sets a no flux condition for the fluid phase
- **Continuity** (under **Fluid** subnode only), which ensures the continuity of the temperature between the fluid phase and the other domains.

Other subnodes implementing boundary conditions specific to the fluid and pellets may be added, to model flow conditions, heat sources, fluxes, and phase change. This

can be done by right-clicking **Fluid** or **Pellets** to select physics features from the context menu.

By default, the shape functions used for the temperature in porous media are **Linear**.

📝	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

🔍	Theory for Heat Transfer in Porous Media

The Heat Transfer in Moist Porous Media Interface

The **Heat Transfer in Moist Porous Media (ht)** interface (![icon]) is automatically added when the **Moist Porous Media** version of the predefined multiphysics interface **Heat and Moisture Transport** is added. A **Moist 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 moist porous media domains corresponds to the convection-diffusion equation with thermodynamic properties averaging models to account for solid matrix, liquid water, and moist air properties. This equation is valid when the temperatures into the porous matrix and the fluid are in equilibrium.

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**: **Moist 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 Moist Porous Media** to select physics features from the context menu.

By default, the shape functions used for the temperature in moist porous media are **Linear**.

	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

	Theory for Heat Transfer in Moist Porous Media

The Heat Transfer in Building Materials Interface

The **Heat Transfer in Building Materials** interface ( ) is automatically added when the **Building Materials** version of the predefined multiphysics interface **Heat and Moisture Transport** is added. A Building Material model is active by default on all domains.

Use this interface 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.

A second order discretization is used by default for the temperature.

📑	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

🔍	Theory for Heat Transfer in Building Materials

📑	See Building Material Library in the *COMSOL Multiphysics Reference Manual* for a description of materials containing the thermal and hygroscopic properties required by some feature nodes of The Heat Transfer in Building Materials Interface.

The Heat Transfer in Moist Air Interface

The **Heat Transfer in Moist Air** interface ( ) is automatically added when the **Moist Air** version of the predefined multiphysics interface **Heat and Moisture Transport** is added. A Moist Air (Heat Transfer Interface) model is active by default on all domains.

It is used to model heat transfer in moist air by convection and diffusion, by using thermodynamics properties defined as a function of the quantity of vapor in the moist air.

When this physics interface is added, the following default nodes are added in the **Model Builder**: **Moist Air**, **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 Moist Air** to select physics features from the context menu.

The settings are the same as for The Heat Transfer in Fluids Interface. This means that a first order discretization is used by default for the temperature.

🗒	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

🔍	Theory for Heat Transfer in Fluids Theory for Heat Transfer in Moist Air

The Bioheat Transfer Interface

The **Bioheat Transfer (ht)** 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.

📑	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.

🔍	• Theory for Bioheat Transfer • Biological Tissue, Bioheat, and Thermal Damage

▥	*Hepatic Tumor Ablation*: Application Library path **Heat_Transfer_Module/Medical_Technology/tumor_ablation**

The Heat Transfer in Shells Interface

The **Heat Transfer in Shells (htlsh)** 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 layered materials represented by boundaries. The interface is active on all boundaries where a layered material is defined, with a **Solid** model active by default. All functionalities for including other boundary contributions, such as surface-to-ambient radiation, are also available.

A layered material can be defined on the boundaries by adding one of the following nodes:

- **Single Layer Material**

- **Layered Material Link**, together with a **Layered Material** node

- **Layered Material Stack**, together with one or several **Layered Material** nodes

- **Material**, with the **Layer thickness** property specified

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

Although the layered material is represented as a boundary in the model, the through-thickness variation of the temperature is accounted for. It means that the temperature equation corresponding to the Fourier's law (see Equation 4-72) is solved also in the layered material's thickness direction. In addition, a single boundary may represent several layers with different thermal properties varying through the thickness of the layered material. This uses the Extra Dimension tool which defines the equations on the product space between the boundary and the additional dimension for the thickness of the layered material. See Modeling Layered Materials for details.

For thermally thin boundaries, it is possible to bypass the use of the product space, by selecting **Nonlayered shell** in the **Shell type** list, and setting a user defined value for the **Thickness** L_{th} directly in the interface. A lumped formulation assuming that heat transfer mainly follows the tangential direction of the boundary is then available.

See Theoretical Background of the Different Formulations for a description of the different formulations.

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**: **Solid**, **Thermal Insulation** (an edge condition), and **Initial Values**. Then, from the **Physics** toolbar, add additional nodes that implement, for example, boundary interface or edge conditions, and heat sources. You can also right-click **Heat Transfer in Shells** to select physics features from the context menu.

BOUNDARY SELECTION

See Boundary Selection for a description this section, common to all versions of the Heat Transfer in Shells interface.

SHELL PROPERTIES

See Shell Properties for a description this section, common to all versions of the Heat Transfer in Shells interface.

PHYSICAL MODEL

See Physical Model for a description of the **Reference temperature** setting under the **Physical Model** section.

CONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button (👁) and selecting **Stabilization** in the **Show More Options** dialog box. See Consistent Stabilization for more details.

INCONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button (👁) and selecting **Stabilization** in the **Show More Options** dialog box. See Inconsistent Stabilization for more details.

DISCRETIZATION

See Discretization for more details.

Temperature
By default, the shape functions used for the temperature are **Quadratic Lagrange**. This setting affects also the discretization of the temperature field in the thickness direction.

See Dependent Variables for details.

🔍	See Settings for the Heat Transfer in Shells Interface for a description of the other settings.

🔍	• Handling Frames in Heat Transfer • Feature Nodes for the Heat Transfer in 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 Shells Interface

This section details the nodes available with The Heat Transfer in Shells Interface with default settings:

• Boundary Nodes for the Heat Transfer in Shells Interface
• Boundary Interface Nodes for the Heat Transfer in Shells Interface
• Edge Nodes for the Heat Transfer in Shells Interface

🔍	Some nodes are only available with some COMSOL products. For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

🔍	The Heat Transfer in Shells Interface does not have any domain node, as it can be applied only on boundaries.

BOUNDARY NODES FOR THE HEAT TRANSFER IN SHELLS INTERFACE

The Heat Transfer in Shells Interface has the following boundary nodes:

- Heat Source (Heat Transfer in Shells Interface)
- Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)

BOUNDARY INTERFACE NODES FOR THE HEAT TRANSFER IN SHELLS INTERFACE

The Heat Transfer in Shells Interface has the following boundary interface nodes:

- Deposited Beam Power, Interface (Heat Transfer in Shells Interface)
- Heat Flux, Interface (Heat Transfer in Shells Interface)
- Heat Source, Interface (Heat Transfer in Shells Interface)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in Shells Interface)

EDGE NODES FOR THE HEAT TRANSFER IN SHELLS INTERFACE

The Heat Transfer in Shells Interface has the following edge nodes:

- Heat Flux (Heat Transfer in Shells Interface)
- Heat Source (Heat Transfer in Shells Interface)
- Heat Flux (Heat Transfer in Shells Interface)
- Lumped System Connector (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

The Heat Transfer in Films Interface

The **Heat Transfer in Films (htlsh)** 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 layered materials represented by boundaries. The interface is active on all boundaries where a layered material is defined, with a **Fluid** model active by default. All functionalities for including other boundary contributions, such as surface-to-ambient radiation, are also available.

> A layered material can be defined on the boundaries by adding one of the following nodes:
>
> - **Single Layer Material**
> - **Layered Material Link**, together with a **Layered Material** node
> - **Layered Material Stack**, together with one or several **Layered Material** nodes
> - **Material**, with the **Layer thickness** property specified
>
> See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

Although the layered material is represented as a boundary in the model, the through-thickness variation of the temperature is accounted for. It means that the convection-diffusion equation is solved also in the layered material's thickness direction. In addition, a single boundary may represent several layers with different thermal properties varying through the thickness of the layered material. This uses the Extra Dimension tool which defines the equations on the product space between the boundary and the additional dimension for the thickness of the layered material. See Modeling Layered Materials for details.

For thermally thin boundaries, it is possible to bypass the use of the product space, by selecting **Nonlayered shell** in the **Shell type** list, and setting a user defined value for the **Thickness** L_{th} directly in the interface. A lumped formulation assuming that heat transfer mainly follows the tangential direction of the boundary is then available.

See Theoretical Background of the Different Formulations for a description of the different formulations.

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**: **Fluid**, **Thermal Insulation** (an edge condition), and **Initial Values**. Then, from the **Physics** toolbar, add additional nodes that implement, for example, boundary interface or edge conditions, and heat sources. You can also right-click **Heat Transfer in Films** to select physics features from the context menu.

BOUNDARY SELECTION

See Boundary Selection for a description this section, common to all versions of the Heat Transfer in Shells interface.

SHELL PROPERTIES

See Shell Properties for a description this section, common to all versions of the Heat Transfer in Shells interface.

PHYSICAL MODEL

See Physical Model for a description of the **Reference temperature** setting under the **Physical Model** section.

CONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button (👁) and selecting **Stabilization** in the **Show More Options** dialog box. See Consistent Stabilization for more details.

INCONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button (👁) and selecting **Stabilization** in the **Show More Options** dialog box. See Inconsistent Stabilization for more details.

DISCRETIZATION

See Discretization for more details.

Temperature
By default, the shape functions used for the temperature are **Linear**. This setting affects also the discretization of the temperature field in the thickness direction.

DEPENDENT VARIABLES

See Dependent Variables for details.

🔍	See Settings for the Heat Transfer in Shells Interface for a description of the other settings.

🔍	• Handling Frames in Heat Transfer • Feature Nodes for the Heat Transfer in Films Interface • Theory for Heat Transfer in Thin Structures

Feature Nodes for the Heat Transfer in Films Interface

This section details the nodes available with The Heat Transfer in Films Interface with default settings:

- Boundary Nodes for the Heat Transfer in Films Interface
- Boundary Interface Nodes for the Heat Transfer in Films Interface
- Edge Nodes for the Heat Transfer in Films Interface

🔍	Some nodes are only available with some COMSOL products. For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

🔍	The Heat Transfer in Films Interface does not have any domain node, as it can be applied only on boundaries.

BOUNDARY NODES FOR THE HEAT TRANSFER IN FILMS INTERFACE

The Heat Transfer in Films Interface has the following boundary nodes:

- Heat Source (Heat Transfer in Shells Interface)
- Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)

BOUNDARY INTERFACE NODES FOR THE HEAT TRANSFER IN FILMS INTERFACE

The Heat Transfer in Films Interface has the following boundary interface nodes:

- Deposited Beam Power, Interface (Heat Transfer in Shells Interface)
- Heat Flux, Interface (Heat Transfer in Shells Interface)
- Heat Source, Interface (Heat Transfer in Shells Interface)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in Shells Interface)

EDGE NODES FOR THE HEAT TRANSFER IN FILMS INTERFACE

The Heat Transfer in Films Interface has the following edge nodes:

- Heat Flux (Heat Transfer in Shells Interface)
- Heat Source (Heat Transfer in Shells Interface)
- Heat Flux (Heat Transfer in Shells Interface)
- Lumped System Connector (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

The Heat Transfer in Fractures Interface

The **Heat Transfer in Fractures (htlsh)** 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 layered materials represented by boundaries. The interface is active on all boundaries where a layered material is defined, with a **Porous Medium** model active by default. All functionalities for including other boundary contributions, such as surface-to-ambient radiation, are also available.

Q

A layered material can be defined on the boundaries by adding one of the following nodes:

- **Single Layer Material**
- **Layered Material Link**, together with a **Layered Material** node
- **Layered Material Stack**, together with one or several **Layered Material** nodes
- **Material**, with the **Layer thickness** property specified

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

Although the layered material is represented as a boundary in the model, the through-thickness variation of the temperature is accounted for. It means that the temperature equation, corresponding to the convection-diffusion equation with thermodynamic properties averaging models to account for both solid matrix and fluid properties, is solved also in the layered material's thickness direction. This 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.

In addition, a single boundary may represent several layers with different thermal properties varying through the thickness of the layered material. This uses the Extra Dimension tool which defines the equations on the product space between the

boundary and the additional dimension for the thickness of the layered material. See Modeling Layered Materials for details.

For thermally thin boundaries, it is possible to bypass the use of the product space, by selecting **Nonlayered shell** in the **Shell type** list, and setting a user defined value for the **Thickness** L_{th} directly in the interface. A lumped formulation assuming that heat transfer mainly follows the tangential direction of the boundary is then available.

See Theoretical Background of the Different Formulations for a description of the different formulations.

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**: **Porous Medium**, **Thermal Insulation** (an edge condition), and **Initial Values**. Then, from the **Physics** toolbar, add additional nodes that implement, for example, boundary interface or edge conditions, and heat sources. You can also right-click **Heat Transfer in Fractures** to select physics features from the context menu.

BOUNDARY SELECTION

See Boundary Selection for a description this section, common to all versions of the Heat Transfer in Shells interface.

SHELL PROPERTIES

See Shell Properties for a description this section, common to all versions of the Heat Transfer in Shells interface.

PHYSICAL MODEL

See Physical Model for a description of the **Reference temperature** setting under the **Physical Model** section.

CONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button ( ) and selecting **Stabilization** in the **Show More Options** dialog box. See Consistent Stabilization for more details.

INCONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button ( ) and selecting **Stabilization** in the **Show More Options** dialog box. See Inconsistent Stabilization for more details.

See Discretization for more details.

Temperature

By default, the shape functions used for the temperature are **Linear**. This setting affects also the discretization of the temperature field in the thickness direction.

DEPENDENT VARIABLES

See Dependent Variables for details.

	See Settings for the Heat Transfer in 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
- Boundary Interface Nodes for the Heat Transfer in Fractures Interface
- Edge 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 https://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:

- Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
- Heat Source (Heat Transfer in Shells Interface)
- Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)

BOUNDARY INTERFACE NODES FOR THE HEAT TRANSFER IN FRACTURES INTERFACE

The Heat Transfer in Fractures Interface has the following boundary interface nodes:

- Deposited Beam Power, Interface (Heat Transfer in Shells Interface)
- Heat Flux, Interface (Heat Transfer in Shells Interface)
- Heat Source, Interface (Heat Transfer in Shells Interface)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in 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):

- Heat Flux (Heat Transfer in Shells Interface)
- Heat Source (Heat Transfer in Shells Interface)
- Heat Flux (Heat Transfer in Shells Interface)
- Lumped System Connector (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

The Lumped Thermal System Interface

The **Lumped Thermal System (lts)** interface ( ) is used to model heat transfer in a system using a thermal network representation.

By analogy with electrical circuits, it provides a lumped element model that idealizes the domain and boundary conditions for heat transfer into components such as thermal resistors, thermal capacitors, and heat rate sources, joined by a network of perfectly thermally conductive wires.

Components representing heat pipes, thermoelectric modules, or user defined thermal subsystems are also available, and the temperatures and heat rates on boundaries can be defined in terminal nodes, including connections to distributed finite element models.

The physics interface supports stationary and time-domain analysis, and solves an energy conservation equation using global variables (space-independent) for the temperatures and heat rates at elements. It is available in all space dimensions.

SETTINGS

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 `lts`.

SYSTEM SETTINGS

Specify the following in this section:

• **Create unique nodes for new components**

• **Check for hanging nodes before solving**

When **Create unique nodes for new components** is selected, newly added components will be assigned unused node names. The components will be disconnected from the rest of the system and the nodes should be updated to reflect the actual system

connections. When this setting is deselected, new components will be connected to the lowest-numbered nodes, starting from 0.

When **Check for hanging nodes before solving** is selected, a check for hanging (not connected) nodes is performed before solving the equations. An error message is displayed if one or more hanging nodes are present in the system.

DEPENDENT VARIABLES

Every feature of the physics interface adds certain ODE dependent variables in the form of temperatures, heat rates, and others. All the ODE dependent variables are grouped in the following ODE fields:

- **Temperatures** (default field name: temperatures)
- **Heat Rates** (default field name: heatRates)
- **Other** (default field name: other)

DISCRETIZATION

All the dependent variables in this physics interface are global (space-independent) and do not need any spatial discretization.

Feature Nodes for the Lumped Thermal System Interface

The **Lumped Thermal System** interface has only global nodes. They are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking then main physics interface node to access the context menu (all users).

📋	Nodes in the lumped thermal system should not be confused with nodes in the Model Builder tree of the COMSOL Multiphysics software. The system node names are not restricted to numerical values but can contain alphanumeric characters.

❗	In the *COMSOL Multiphysics Reference Manual* see Table 2-4 for links to common sections and Table 2-5 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 following nodes are available from the **Physics** toolbar, **Global** menu or from by right-clicking the **Lumped Thermal System** interface node.

- Conductive Thermal Resistor
- Convective Thermal Resistor
- Heat Pipe
- Heat Rate Source
- Radiative Thermal Resistor
- Thermal Capacitor
- Thermoelectric Module

The following nodes are available from the **Physics** toolbar, **Global** menu or from by selecting from the **Subsystems** subnode after right-clicking the **Lumped Thermal System** interface node.

- Subsystem Definition
- Subsystem Instance

The following nodes are available from the **Physics** toolbar, **Global** menu or from by selecting from the **Nodes** subnode after right-clicking the **Lumped Thermal System** interface node.

- External Terminal
- Heat Rate
- Radiative Heat Rate
- Temperature
- Thermal Mass

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 (transparent medium).

> 📄 This interface does not compute the temperature field, it requires it as model input. If the medium participates in the radiation (semitransparent medium), then use The Radiation in Participating Media Interface instead.

The radiosity method is used on boundaries where **Surface-to-Surface Radiation** is enabled.

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. **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

Define the **Wavelength dependence of radiative properties**.

- Keep the default value, **Constant,** to define a diffuse gray radiation model. In this case, the surface properties (emissivity, radiosity, reflectivity, transmissivity, critical angle) have the same definition for all wavelengths. The surface properties 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, $[0, \lambda_{sol/amb}]$, (solar radiation) and one for large wavelengths, $[\lambda_{sol/amb}, +\infty[$, (ambient radiation). It is then possible to define the **Separation point between spectral bands** (SI unit: m), $\lambda_{sol/amb}$, to adjust the wavelength intervals corresponding to the solar and ambient radiation. The surface properties can 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 value of the **Right endpoint** for each spectral band in the table, to define a diffuse spectral radiation model. The value of the **Right endpoint** must be entered without unit. Modify the **Wavelength unit** to set the unit of all the endpoints. **Right endpoint** values should be set in an ascending order. The values of the **Left endpoint** for the next spectral band are updated automatically. It is possible to provide a definition of the surface properties for each spectral band.

📝	The first **Left endpoint** and the last **Right endpoint** are predefined and equal to 0 and +∞, respectively.
	Parameters can be used to set the value of the **Right endpoint**.

❗	The wavelengths λ set in **Solar and ambient** and **Multiple spectral bands** are the wavelengths in vacuum.

Modify the **Transparent media refractive index** to define the refractive index. Vacuum has a refractive index of 1, which is usually a good approximation for air refractive index.

Select the **Use radiation groups** check box to be able to define Radiation Group (Surface-to-Surface Radiation Interface). Radiation groups can be used to speed up the radiation calculations by gathering boundaries that can see one another.

Select the **Surface-to-surface radiation method**: **Direct area integration**, **Hemicube** (the default), or **Ray shooting**:

• For Direct Area Integration select a **Radiation integration order** — **4** is the default.

• For Hemicube select a **Radiation resolution** — **256** is the default.

• For Ray Shooting select a **Radiation resolution** — **8** is the default.

TABLE 5-4: SURFACE-TO-SURFACE RADIATION METHODS

	DIRECT AREA INTEGRATION	HEMICUBE	RAY SHOOTING
Supported dimensions	3D, 2Daxi, 2D, 1D axi,1D	3D, 2Daxi, 2D, 1D axi,1D	3D, 2D, 2Daxi
Shadowing effects		√	√
Diffuse reflection	√	√	√
Specular reflection			√
Refraction			√
Directional properties			√
Wavelength dependent properties	√ (Solar and ambient / Multiple spectral bands)	√ (Solar and ambient / Multiple spectral bands)	√ (Solar and ambient / Multiple spectral bands)
External Radiation Source	√	√	√
Symmetry	√	√	√
View factor update control	√	√	√

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 radiation 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.

Ray Shooting

The ray shooting algorithm accounts for directional properties in the view factor evaluation. This includes specular reflection and transmission of radiation and also the

angular dependence of surface properties. Select this method to enable the Opaque Surface (Surface-to-Surface Radiation Interface) and Semitransparent Surface (Surface-to-Surface Radiation Interface) boundary conditions.

To compute the radiation incident upon surfaces, the ray shooting algorithm uses a backwards ray tracing approach, emitting rays from each element outwards, to determine the total irradiation from surrounding elements. For each element, n+1 rays are launched in 2D and n^2+1 rays in 3D where n is the value selected for **Radiation resolution**. To determine the direction of these rays, the hemisphere (angular space) is divided into n tiles in 2D and n^2 in 3D. The trajectories are computed as the rays are absorbed, reflected or transmitted on the model surfaces until their intensity becomes too small or if the rays go far away from the geometry. The threshold where the ray trajectory is no longer computed is controlled by the **Tolerance**. On their trajectory, two adjacent rays can hit boundaries belonging to two different features, or detects differences in the radiosity. In such cases, the number of tiles can be locally refined up to a number of times defined by the **Maximum number of adaptations**. This aims to detect irradiation from boundaries that are smaller than the tile size.

Set the **Maximal number of reflections** (default value is 1000) to define the maximal number of times that a specific ray is allowed to be reflected by a surface. This avoids infinite loops in case of perfectly reflecting surfaces. Using large values improves the view factor accuracy while small ones can be used to limit the view factor evaluation time.

When using directional dependent surface properties, set **Angular dependent properties** to define how the selected directional functions are evaluated.

- Keep the default value, **Full resolution,** to evaluate the directional function for each direction. This option improves accuracy and is the most computationally intensive.

- Select **Interpolated function** to sample the directional function on a grid with (k+1) points for Polar functions and $(k+1)^2$ grid points for Polar and Azimuthal functions. k is the **Number of interpolation points** (default value is 100), a strictly positive integer. The evaluation is done using linear interpolation between the sampled values. This option can speed up the view factor evaluation when directional dependent surface properties are used.

To improve the accuracy of the radiation computation the user may increase the **Radiation resolution** (default value is 8), decrease the **Tolerance** (default value is 1e-3) or increase the **Maximum number of adaptations** (default value is 3). Conversely changing these values in the opposite direction should decrease computational time.

Also, higher values of the **Geometry shape order** under **Component** node may improve the results.

For an axisymmetric geometry, G_m and F_{amb} must be evaluated in a corresponding 3D geometry. The number of bounces of the rays in the azimuthal direction can be controlled by setting the **Azimuthal discretization** (default value is 100).

VIEW FACTOR

This section is available by clicking the **Show More Options** button ( ) and selecting **Advanced Physics Options** in the **Show More Options** dialog box.

Select the **Store view factors on disk** check box to store the view factors (mutual and external irradiation) in the model after the next computation. View factors are reused automatically provided the mesh, the selection of the radiating boundaries, the radiation direction, external radiation sources or symmetry conditions are unchanged. However, adding or removing a feature or modifying a feature selection in the **Surface-to-Surface radiation** always forces to recompute view factors. The view factors are reused for example if only the value of the surface emissivity is changed. When coupled to a **Heat Transfer** interface through a **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling, changes for example on temperature, source or flux conditions can be made in the **Heat Transfer** interface without having to recompute the view factors. The view factors are stored in the model file which may induce a noticeable increase in file size. When this check box is cleared and the model is saved, view factors are removed from the model file. View factors are also removed from the model when clearing the mesh and the solution.

Define the **View factors update threshold** to control the frequency of view factors computation in time-dependent simulation. This setting is used in models containing a **Moving Mesh** node under **Definitions**, when the **Symmetry for Surface-to-Surface Radiation** feature uses one or more moving symmetry planes or with time-dependent specular surface properties.

- Keep the default value, **Every nonlinear iteration,** to update the view factors every nonlinear iteration. This option is the most computationally intensive.

- Select **Every Nth time** to update the view factors at least every Nth time where N is the **View factors update time** (SI unit: s). For example, set **View factors update time** to 1e-3[s] to enforce the update every 1e-3s of simulation time.

- Select **Never update** to never update view factors. This option can be used when view factors update has little effect on the results (short timescales, small geometry changes). This option is the least computationally intensive.

- Select **User defined** to define the view factors update threshold manually. The view factors are updated each time the **Expression** changes by more than the **Tolerance**. By default, the tolerance is set to -1 to force the view factor to be recomputed every nonlinear iteration. For example, set **Expression** to `intop1(1)` and **Tolerance** to `0.15[m]`, where `intop1()` is an integration operator over a deforming boundary, to update view factors when the boundary length changes more than 0.15m compared with the previous view factors calculation.

Select the **High order mesh elements** check box to use the mesh elements as they are defined by the geometry shape functions to compute view factors. This option is designed to improve the accuracy and is expected to be more computationally expensive. If not selected, the view factors are computed using linearized mesh elements.

Select the **Mesh element self-irradiation** to account for self-irradiation when a mesh element is concave with respect to the radiation direction. Self-irradiation is always accounted for in ray shooting.

JACOBIAN CONTRIBUTION

This section is available by clicking the **Show More Options** button (👁) and selecting **Advanced Physics Options** in the **Show More Options** dialog box.

Set the **Jacobian contribution**.

- Keep the default value, **Only local contributions to radiosity,** to exclude the irradiation contribution to the Jacobian matrix block corresponding to the radiosity degrees of freedom. This option is expected to reduce dramatically the time and the memory required to solve the model.

- Select **Include contributions from total irradiation** to include the irradiation in the Jacobian matrix and perform the full Jacobian evaluation. This option can ease the convergence.

Provided a converged solution is obtained with both options, the same solution is obtained, within the solver tolerance.

Surface Radiosity

Select **Linear** (the default), **Quadratic**, **Cubic**, **Quartic** or **Quintic** to define the discretization level used for the **Surface radiosity** shape function.

Q
- 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

---

Q
Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit https://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 (Surface-to-Surface Radiation Interface).

BOUNDARY NODES FOR THE SURFACE-TO-SURFACE RADIATION INTERFACE

The Surface-to-Surface Radiation Interface has the following boundary nodes:

- Diffuse Mirror (Surface-to-Surface Radiation Interface)
- Diffuse Surface (Surface-to-Surface Radiation Interface)
- External Radiation Source
- Opaque Surface (Surface-to-Surface Radiation Interface)
- Prescribed Radiosity (Surface-to-Surface Radiation Interface)
- Semitransparent Surface (Surface-to-Surface Radiation Interface)
- Radiation Group (Surface-to-Surface Radiation Interface)

The Radiation in Participating Media Interface

The **Radiation in Participating Media (rpm)** interface (![icon]), found under the **Heat Transfer>Radiation** branch (![icon]), is used to model radiative heat transfer inside participating (semitransparent media) media.

> ![note icon]
> This physics interface doesn't compute the temperature field and 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**: **Participating Medium** and **Opaque Surface**. In addition, when **Discrete ordinates method** is selected, **Continuity on Interior Boundary** is also added. 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

Wavelength Dependence of Radiative Properties

Define the **Wavelength dependence of radiative properties**:

- Keep the default value, **Constant,** to define a gray radiation model. In this case, the radiative properties (emissivity, absorption and scattering coefficients) have the same definition for all wavelengths. These properties can still depend on other quantities, in particular on the temperature.

- Select **Solar and ambient** to define a spectral radiation model with two spectral bands, one for short wavelengths, $[0, \lambda_{sol/amb}]$, (solar radiation) and one for large wavelengths, $[\lambda_{sol/amb}, +\infty[$, (ambient radiation). It is then possible to define the **Separation point between spectral bands** (SI unit: m), $\lambda_{sol/amb}$, to adjust the wavelength intervals corresponding to the solar and ambient radiation. The radiative properties can then be defined for each spectral band.

- Select **Multiple spectral bands** and set the value of the **Right endpoint** for each spectral band in the table, to define a spectral radiation model. The value of the **Right endpoint** must be entered without unit. Modify the **Wavelength unit** to set the unit of all the endpoints. **Right endpoint** values should be set in an ascending order. The values of the **Left endpoint** for the next spectral band are updated automatically. It is possible to provide a definition of the surface properties for each spectral band.

🗎	The first **Left endpoint** and the last **Right endpoint** are predefined and equal to 0 and $+\infty$, respectively.
	Parameters can be used to set the value of the **Right endpoint**.

❗	The wavelengths λ set in **Solar and ambient** and **Multiple spectral bands** are the wavelengths in vacuum.

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default), or **P1 approximation**.

- When **Discrete ordinates method** is selected, Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) and Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) are automatically added as default features.

- When **P1 approximation** is selected, Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) is automatically added as a default feature and both this and Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) 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 (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) is not available.

| ! | The choice of **Radiation discretization method** also offers different settings for the Participating Medium (Radiation in Participating Media Interface) (all methods), Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) (P1 approximation), and Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) (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.

Quadrature Set

When **Discrete ordinates method** is selected, the method of angular discretization of the radiative intensity direction should be specified. In particular, it defines the method of computation of the weights w_j used in the approximation of the scattering term:

$$\int_{4\pi} I(\Omega)d\Omega \approx \sum_{j=1}^{n} w_j I_j$$

Choose a **Quadrature set** from the list:

- The **Level Symmetric Even** (the default), **Level Symmetric Hybrid**, and **Equal Weight Odd** sets are S_N approximations. Depending on the set and the order selected in the **Discrete ordinates method** list, different moment conditions are satisfied. In 3D, S2, S4, S6, and S8 generate 8, 24, 48, and 80 directions, respectively. In 2D, S2, S4, S6, and S8 generate 4, 12, 24, and 40 directions, respectively.

- The **Quasi-uniform** weight approximation discretizes the angular space by using a reference octahedron with 8 triangular faces, further discretized in function of the order of the method. This corresponds to a T_N approximation, for which $8N^2$ weights are computed at order N.

> 🔍 See Discrete Ordinates Method (DOM) for details about the S_N and T_N quadrature sets.

DISCRETIZATION

Radiative Intensity
When the **Radiation discretization method** is set to **Discrete ordinates method** in **Participating Media Settings**, set the discretization level of the **Radiative intensity**: **Constant**, **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

Incident Radiation
When the **Radiation discretization method** is set to **P1 approximation** in **Participating Media Settings**, set the discretization level of the **Incident radiation**: **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

> ℚ Some nodes are only available with some COMSOL products.
>
> For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE RADIATION IN PARTICIPATING MEDIA INTERFACE

The Radiation in Participating Media Interface has the following domain nodes:

• Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

• Participating Medium (Radiation in Participating Media Interface)

• Radiative Source

BOUNDARY NODES FOR THE RADIATION IN PARTICIPATING MEDIA INTERFACE

The Radiation in Participating Media Interface has the following boundary nodes:

• Continuity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

• Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

• Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

• Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

The Radiation in Absorbing-Scattering Media Interface

The **Radiation in Absorbing-Scattering Media (rasm)** interface ( ), found under the **Heat Transfer>Radiation** branch ( ), is used to model radiative heat transfer inside participating (semitransparent media) media supposed to be non-emitting.

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**: **Absorbing-Scattering Medium** and **Opaque Surface**. In addition, when **Discrete ordinates method** is selected, **Continuity on Interior Boundary** is also added. 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 Absorbing-Scattering Media** node to select physics features from the context menu.

In this section:

- Settings for the Radiation in Absorbing-Scattering Media Interface
- Feature Nodes for the Radiation in Absorbing-Scattering Media Interface

Settings for the Radiation in Absorbing-Scattering 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 `rasm`.

Wavelength Dependence of Radiative Properties

Define the **Wavelength dependence of radiative properties**:

- Keep the default value, **Constant,** to define a gray radiation model. In this case, the radiative properties (emissivity, absorption and scattering coefficients) have the same definition for all wavelengths. These properties can still depend on other quantities, in particular on the temperature.

- Select **Solar and ambient** to define a spectral radiation model with two spectral bands, one for short wavelengths, $[0, \lambda_{sol/amb}]$, (solar radiation) and one for large wavelengths, $[\lambda_{sol/amb}, +\infty[$, (ambient radiation). It is then possible to define the **Separation point between spectral bands** (SI unit: m), $\lambda_{sol/amb}$, to adjust the wavelength intervals corresponding to the solar and ambient radiation. The radiative properties can then be defined for each spectral band.

- Select **Multiple spectral bands** and set the value of the **Right endpoint** for each spectral band in the table, to define a spectral radiation model. The value of the **Right endpoint** must be entered without unit. Modify the **Wavelength unit** to set the unit of all the endpoints. **Right endpoint** values should be set in an ascending order. The values of the **Left endpoint** for the next spectral band are updated automatically. It is possible to provide a definition of the surface properties for each spectral band.

🗐	The first **Left endpoint** and the last **Right endpoint** are predefined and equal to 0 and +∞, respectively.
	Parameters can be used to set the value of the **Right endpoint**.

⚠	The wavelengths λ set in **Solar and ambient** and **Multiple spectral bands** are the wavelengths in vacuum.

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default), or **P1 approximation**.

- When **Discrete ordinates method** is selected, Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) and Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) are automatically added as default boundary features. The Incident Intensity (Radiation in Participating Media and

Radiation in Absorbing-Scattering Media Interfaces) and Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) features are made available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or the context menu (all users).

- When **P1 approximation** is selected, Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) is automatically added as a default boundary feature. The Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces), Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces), and Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) features 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 (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) is not available.

> ⚠️ The choice of **Radiation discretization method** also offers different settings for the Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface), Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces), and Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) 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.

Quadrature Set

When **Discrete ordinates method** is selected, the method of angular discretization of the radiative intensity direction should be specified. In particular, it defines the method of computation of the weights w_j used in the approximation of the scattering term:

$$\int_{4\pi} I(\Omega)d\Omega \approx \sum_{j=1}^{n} w_j I_j$$

Choose a **Quadrature set** from the list:

- The **Level Symmetric Even** (the default), **Level Symmetric Hybrid**, and **Equal Weight Odd** sets are S_N approximations. Depending on the set and the order selected in the **Discrete ordinates method** list, different moment conditions are satisfied. In 3D, S2, S4, S6, and S8 generate 8, 24, 48, and 80 directions, respectively. In 2D, S2, S4, S6, and S8 generate 4, 12, 24, and 40 directions, respectively.

- The **Quasi-uniform** weight approximation discretizes the angular space by using a reference octahedron with 8 triangular faces, further discretized in function of the order of the method. This corresponds to a T_N approximation, for which $8N^2$ weights are computed at order N.

> 🔍 See Discrete Ordinates Method (DOM) for details about the S_N and T_N quadrature sets.

DISCRETIZATION

Radiative Intensity
When the **Radiation discretization method** is set to **Discrete ordinates method** in **Participating Media Settings**, set the discretization level of the **Radiative intensity**: **Constant**, **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

Incident Radiation
When the **Radiation discretization method** is set to **P1 approximation** in **Participating Media Settings**, set the discretization level of the **Incident radiation**: **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

Feature Nodes for the Radiation in Absorbing-Scattering Media Interface

This section details the nodes available with The Radiation in Absorbing-Scattering Media Interface with default settings:

- Domain Nodes for the Radiation in Absorbing-Scattering Media Interface

- Boundary Nodes for the Radiation in Absorbing-Scattering Media Interface

---

🔍   Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE RADIATION IN ABSORBING-SCATTERING MEDIA INTERFACE

The Radiation in Absorbing-Scattering Media Interface has the following domain nodes:

- Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface)
- Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Radiative Source

BOUNDARY NODES FOR THE RADIATION IN ABSORBING-SCATTERING MEDIA INTERFACE

The Radiation in Absorbing-Scattering Media Interface has the following boundary nodes:

- Continuity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

The Radiative Beam in Absorbing Media Interface

The **Radiative Beam in Absorbing Media (rbam)** interface ( ), found under the **Heat Transfer>Radiation** branch ( ), is used to model the attenuation of an incident light within a semitransparent material due to absorption. The beam intensity is determined using the Beer-Lambert Law. As the beam is absorbed it deposits energy which acts as a heat source.

When the physics interface is added, these default nodes are added to the **Model Builder**: **Absorbing Medium**, **Transparent Surface**, 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 **Radiative Beam in Absorbing Media** node to select physics features from the context menu.

In this section:

- Settings for the Radiative Beam in Absorbing Media Interface
- Feature Nodes for the Radiative Beam in Absorbing Media Interface

Settings for the Radiative Beam in Absorbing 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 rbam.

	For additional background theory also see Radiative Beam in Absorbing Media Theory.

CONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button (⦿) and selecting **Stabilization**. The **Streamline diffusion** check box is selected by default and should remain selected for optimal performance. **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

This section is available by clicking the **Show More Options** button (⦿) and selecting **Stabilization** in the **Show More Options** dialog box. The **Isotropic diffusion** check box is not selected by default.

DISCRETIZATION

Intensity
Set the discretization level of the intensity: **Constant**, **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic**.

Feature Nodes for the Radiative Beam in Absorbing Media Interface

This section details the nodes available with The Radiative Beam in Absorbing Media Interface with default settings:

• Domain Nodes for the Radiative Beam in Absorbing Media Interface
• Boundary Nodes for the Radiative Beam in Absorbing Media Interface

---

Q | Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

DOMAIN NODES FOR THE RADIATIVE BEAM IN ABSORBING MEDIA INTERFACE
The Radiative Beam in Absorbing Media Interface has the following domain nodes:

• Initial Values (Radiative Beam in Absorbing Medium Interface)
• Absorbing Medium (Radiative Beam in Absorbing Medium Interface)

BOUNDARY NODES FOR THE RADIATIVE BEAM IN ABSORBING MEDIA INTERFACE

The Radiative Beam in Absorbing Media Interface has the following boundary nodes:

- Incident Intensity (Radiative Beam in Absorbing Medium Interface)
- Opaque Surface (Radiative Beam in Absorbing Medium Interface)
- Transparent Surface (Radiative Beam in Absorbing Medium Interface)

The Moisture Transport in Air Interface

The **Moisture Transport in Air (mt)** interface (⬡) is used to model moisture transport through convection and diffusion of vapor in moist air. A Moist Air (Moisture Transport Interface) model is active by default on all domains.

When this physics interface is added, these default nodes are added to the **Model Builder**: **Moist Air**, **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 in Air** to select physics features from the context menu.

By default, the shape functions used for the relative humidity in porous media are **Linear**.

📄	See Settings for the Moisture Transport Interface and Feature Nodes for the Moisture Transport Interface for details.

🔍	Theory for Moisture Transport

The Moisture Transport in Porous Media Interface

The **Moisture Transport in Porous Media (mt)** interface ( ) is used to model moisture transport into the pores of a porous medium, through convection and diffusion of vapor in moist air, and convection and capillary flow of liquid water. An Hygroscopic 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**: **Hygroscopic 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 in Porous Media** to select physics features from the context menu.

By default, the shape functions used for the relative humidity in porous media are **Linear**.

📄	See Settings for the Moisture Transport Interface and Feature Nodes for the Moisture Transport Interface for details.

🔍	Theory for Moisture Transport

The Moisture Transport in Building Materials Interface

The **Moisture Transport in Building Materials (mt)** interface ( ) is used to model moisture transport in building materials by taking into account moisture storage, liquid transport by capillary suction forces, and the diffusive transport of vapor. A Building Material model is active by default on all domains.

When this physics interface is added, these default nodes are added to the **Model Builder**: **Building Material**, **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 in Building Materials** to select physics features from the context menu.

The shape functions used for the relative humidity are **Quadratic Lagrange**.

📄	See Building Material Library in the *COMSOL Multiphysics Reference Manual* for a description of materials containing the thermal and hygroscopic properties required by some feature nodes of the Moisture Transport in Building Materials Interface.

📄	See Settings for the Moisture Transport Interface and Feature Nodes for the Moisture Transport Interface for details.

🔍	Theory for Moisture Transport

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 (macOS 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
- Boundary Features
- Boundary Interface Features
- Edge Features
- Point Features
- Global Features

Domain Features

The following domain nodes are available for the Heat Transfer interfaces:

- Absorbing Medium (Radiative Beam in Absorbing Medium Interface)
- Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface)
- Bioheat
- Biological Tissue
- Building Material
- Convectively Enhanced Conductivity
- Cross Section
- Fluid
- Fluid (Porous Medium)
- Geothermal Heating
- Heat Source
- Immobile Fluids (Porous Medium)
- Initial Values
- Initial Values (Radiative Beam in Absorbing Medium Interface)
- Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Irreversible Transformation
- Isothermal Domain
- Liquid Water (Moist Porous Medium)

- Liquid Water (Moist Porous Medium)
- Moist Air (Heat Transfer Interface)
- Moist Air (Moist Porous Medium)
- Moist Porous Medium
- Opacity (Surface-to-Surface Radiation Interface)
- Optically Thick Participating Medium
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Participating Medium (Radiation in Participating Media Interface)
- Pellets (Porous Medium)
- Pellet-Fluid Interface (Porous Medium)
- Phase Change Material
- Porous Matrix (Porous Medium, Moist Porous Medium)
- Porous Medium
- Pressure Work
- Radiative Source
- Shape Memory Alloy
- Solid
- Thermal Damage
- Thermal Dispersion
- Thermoelastic Damping
- Thickness
- Translational Motion
- Viscous Dissipation

For a detailed overview of the functionality available in each product, visit
https://www.comsol.com/products/specifications/

Absorbing Medium (Radiative Beam in Absorbing Medium Interface)

This node calculates the radiative intensity of incident beams propagating in an absorbing medium as the solution of the Beer-Lambert equation:

$$\frac{\mathbf{e}_i}{\|\mathbf{e}_i\|} \cdot \nabla I_i = -\kappa I_i$$

where

- $\mathbf{e}_i$ is the orientation of the ith beam (dimensionless).
- I_i is the intensity of the ith beam (SI unit: W/m^2).
- κ is the absorption coefficient (SI unit: m^{-1}) of the medium.

The energy deposited by the beams propagation in the absorbing medium is calculated and the radiative heat source term Q (SI unit: W/m^3) is defined by:

$$Q = \sum_i \kappa I_i$$

See Incident Intensity (Radiative Beam in Absorbing Medium Interface) for the definition of the orientation and profile of the ith beam.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining 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. The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source**

button (⬚), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

The **Absorption coefficient** of the medium, κ (SI unit: m^{-1}), should be set in this section.

🔍   Radiative Beam in Absorbing Media Theory

LOCATION IN USER INTERFACE

Context Menus
Radiative Beam in Absorbing Media>Absorbing Medium

Ribbon
Physics tab with **Radiative Beam in Absorbing Media** selected:

Domains>Absorbing Medium

Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media 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 calculates the heating due to the propagation of the rays, and takes into account the absorbed and the scattered radiation, depending on the **Radiation discretization method** selected in the Participating Media Settings section of the interface. In particular, it can be used for the modeling of light diffusion in a non-emitting medium.

It calculates the radiative heat source term Q_r (SI unit: W/m^3), defined by:

$$Q_r = \kappa G$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2), defined by

$$G = \int_{4\pi} I(\Omega) d\Omega$$

where

- $I(\Omega)$ 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* with no emission term

$$\Omega \cdot \nabla I(\Omega) = -\beta I(\Omega) + \frac{\sigma_s}{4\pi}\int_0^{4\pi} I(\Omega')\phi(\Omega', \Omega)d\Omega'$$

- $\beta = \kappa + \sigma_s$ 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)

If the **Radiation discretization method** is **Discrete ordinates method**, G is defined by

$$G = \sum_{i=1}^{N} \omega_i I_i$$

and

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi}\sum_{j=1}^{N} \omega_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \tag{6-1}$$

where

- $\mathbf{S}_i$ is the ith discrete ordinate.
- I_i is the ith component of the radiative intensity.
- ω_j is the ith quadrature weight.

If the **Radiation discretization method** is **P1 approximation**, the incident radiation G is the solution of the following equation

$$\nabla \cdot (D_{\text{P1}}\nabla G) - \kappa G = 0 \tag{6-2}$$

where D_{P1} is the P1 diffusion coefficient.

When **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands** in the **Participating Media Settings** section of the interface, Equation 6-1 and Equation 6-2 are solved for each spectral band k: $I_{i,k}$ is the ith component of the radiative intensity for spectral band k, and G_k is the incident radiation for spectral band k.

The characteristics of the two available radiation discretization methods are summarized in the following table.

TABLE 6-1: DISCRETIZATION METHODS FOR RADIATION IN ABSORBING-SCATTERING MEDIA (RASM INTERFACE)

OPTION	DOM	P1
Optical thickness validity	All	$\tau >> 1$
Absorption modeling	Yes	Yes
Scattering modeling	Linear Polynomial	Linear
Computational cost	High: up to 80 additional degrees of freedom (I_i)	Medium: 1 additional degree of freedom (G)

If radiative emission should be considered as well, the Participating Medium (Radiation in Participating Media Interface) node should be used instead. The table below describes the different effects accounted for by the interfaces found under the **Heat Transfer>Radiation** branch ( ).

TABLE 6-2: RADIATION EFFECTS CALCULATED BY THE RADIATION INTERFACES

RADIATION EFFECT	RPM	RASM	RBAM
Absorption	Yes	Yes	Yes
Scattering	Yes	Yes	No
Emission	Yes	No	No

MODEL INPUT

This section contains fields and values that are inputs for expressions defining 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. The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics

interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button ( ), and in the **Default model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

ABSORPTION

The **Absorption coefficient** κ should be specified. It defines the amount of radiation, $\kappa I(\Omega)$, that is absorbed by the medium.

The **Absorption coefficient** κ (SI unit: 1/m) uses values **From material** by default.

For **User defined**, set a value or expression. You can define a temperature-dependent absorption coefficient using the variable `rasm.T`.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength may be accessed through the `rasm.lambda` variable. Any expression set for the absorption coefficient is then averaged on each spectral band to obtain a piecewise constant absorption coefficient. If the average value of the absorption coefficient on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

When **Absorption coefficient** is set to **User defined for each band**, enter a value for the **Absorption coefficient** for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

SCATTERING

This section defines the scattering property of the participating medium. The **Scattering coefficient** σ_s should be specified.

The **Scattering coefficient** σ_s (SI unit: 1/m) uses values **From material** by default.

For **User defined**, set a value or expression. You can define a temperature-dependent scattering coefficient using the variable `rasm.T`.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength may be accessed through the `rasm.lambda` variable. Any expression set for the scattering coefficient is then averaged on each spectral band to obtain a piecewise constant scattering coefficient. If the average value of the scattering coefficient on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

When **Scattering coefficient** is set to **User defined for each band**, enter a value for the **Scattering coefficient** for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

Choose in addition the **Scattering type**: **Isotropic**, **Linear anisotropic**, **Polynomial anisotropic** (only with **Discrete ordinates method**), or **Henyey-Greenstein** (only with **Discrete ordinates method**).

This setting provides options to approximate the scattering phase function ϕ using the cosine of the scattering angle, μ_0:

- **Isotropic** (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.

- For **Henyey-Greenstein** it defines the scattering phase function as

$$\phi(\mu_0) = \frac{1}{K} \cdot \frac{1 - \eta^2}{(1 + \eta^2 - 2\eta\mu_0)^{3/2}}$$

where $-1 < \eta < 1$ is the anisotropy parameter and K is defined as follows to produce a normalized phase function:

$$K = \frac{1}{4\pi} \cdot \int_{4\pi} \frac{1 - \eta^2}{(1 + \eta^2 - 2\eta\mu_0)^{3/2}} d\Omega$$

For **Linear anisotropic** and **Polynomial anisotropic**, select the **Normalize phase function** check box to define a phase function such as

$$\int_{4\pi} \phi(\mu_0) d\Omega = 4\pi$$

The normalization is automatically applied for the **Henyey-Greenstein** option.

For **Linear anisotropic**, **Polynomial anisotropic**, and **Henyey-Greenstein**, select the **Wavelength-dependent scattering type** check box to average each **Legendre coefficient**

$a_1, \ldots, a_{12}$ or the **Anisotropy parameter** μ to obtain piecewise constant coefficients on each spectral band.

Q
- Theory for Radiation in Participating Media
- Discrete Ordinates Method (DOM)
- P1 Approximation Theory
- Radiation in Absorbing-Scattering Media Theory

LOCATION IN USER INTERFACE

Context Menus
Radiation in Absorbing-Scattering Media>Absorbing-Scattering Medium

Ribbon
Physics tab with **Radiation in Absorbing-Scattering Media** selected:

Domains>Absorbing-Scattering Medium

Bioheat

This feature provides the source terms that represent blood perfusion and metabolism to model heat transfer in biological tissue using the bioheat equation:

$$\rho_b C_{p, b} \omega_b (T_b - T) + Q_{met}$$

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.

⊙	• Theory for Bioheat Transfer • Biological Tissue • Thermal Damage

▥	*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

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 4-37. Optionally a damage model can be defined to take into account overheating or freezing in tissues, by adding a Thermal Damage subnode.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button ($\equiv_\uparrow$), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

⚠️ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button ($\equiv_\uparrow$), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

HEAT CONDUCTION, SOLID

The default **Thermal conductivity** k uses values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity and enter another value or expression in the field or matrix.

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics physics interface for example), you can select on which frame the thermal conductivity is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables ht.kxx, ht.kyy, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables ht.kxx, ht.kyy, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{(x, y, z)} = \mathbf{R} k_{(X, Y, Z)} \mathbf{R}^{\mathrm{T}}$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

THERMODYNAMICS, SOLID

The default **Density** ρ and **Heat capacity at constant pressure** C_p are taken **From material**. See Material Density in Features Defined in the Material Frame if a temperature-dependent density should be set.

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.

🔍	• Theory for Bioheat Transfer • Bioheat • Thermal Damage

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

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Specific Media>Biological Tissue
Heat Transfer in Solids>Specific Media>Biological Tissue
Heat Transfer in Fluids>Specific Media>Biological Tissue
Heat Transfer in Porous Media>Specific Media>Biological Tissue
Heat Transfer in Building Materials>Specific Media>Biological Tissue
Bioheat Transfer>Specific Media>Biological Tissue

Ribbon
Physics tab with **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Domains>Specific Media>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 change in material properties due to moisture transfer is accounted for by the change in an apparent thermal conductivity and a heat source or sink expressed in the divergence operator. It takes into account the latent heat of evaporation:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = Q \tag{6-3}$$

$$\mathbf{q} = -(k_{\text{eff}} \nabla T + L_v \delta_p \nabla(\phi_w p_{\text{sat}})) \tag{6-4}$$

with the following material properties, fields, and source:

- $(\rho C_p)_{\text{eff}}$ (SI unit: J/(m^3·K)) is the effective volumetric heat capacity at constant pressure.
- k_{eff} (SI unit: W/(m·K)) is the effective thermal conductivity (a scalar or 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_{w} (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 (or sink). Add one or more heat sources as separate physics features. See the Heat Source node, for example.

For a steady-state problem the temperature does not change with time and the first term disappears.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable minput.Tempref, which is set to 293.15 K by default. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

> ⚠ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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** (![icon]). The available options are: **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Relative Humidity

This section contains an input for the definition of the relative humidity, which is used in the right hand side of Equation 6-4.

The default **Relative humidity** ϕ_w 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. The **Common model input** option is the variable `minput.phi`, set to 0 by default. To edit it, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Relative humidity** in the **Expression for remaining selection** section.

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** (![icon]).

HEAT CONDUCTION

This section provides two options for defining the effective thermal conductivity k_{eff}:

- When **Equivalent thermal conductivity** is selected (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 **Full** 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 as a function of the properties of the solid matrix and moisture:

$$k_{\text{eff}} = k_{\text{s}}\left(1 + \frac{b\,w(\phi_{\text{w}})}{\rho_{\text{s}}}\right)$$

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

The **Dry solid thermal conductivity** k_{s} (SI unit: W/(m·K)) and the **Thermal conductivity supplement** b (dimensionless) should be specified. By default, the **Dry solid thermal conductivity** and **Thermal conductivity supplement** are taken **From material**. For **User defined**, enter values or expressions in the editable fields.

The **Density** ρ_{s} and the **Moisture storage function** $w(\phi_{\text{w}})$ are specified in the Thermodynamics, Dry Solid and Building Material Properties sections respectively.

THERMODYNAMICS, DRY SOLID

This section defines the thermodynamics properties of the dry solid.

The specific heat capacity describes the amount of thermal 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** $C_{p,\text{s}}$ should be specified. The default **Density** and **Specific heat capacity** are taken **From material**. For **User defined**, enter values or expressions in the editable fields.

The effective volumetric heat capacity at constant pressure is defined to take into account both the properties of the solid matrix and the moisture:

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

where

- ρ_{s} (SI unit: kg/m^3) is the density of the dry solid.
- $C_{p,\text{s}}$ (SI unit: J/(kg·K)) is the specific heat capacity of the dry solid.
- $w(\phi_{\text{w}})$ (SI unit: kg/m^3) is the water content given by a moisture storage function.
- $C_{p,\text{w}}$ (SI unit: J/(kg·K)) is the heat capacity of water at constant pressure.

BUILDING MATERIAL PROPERTIES

This section defines the properties of the building material for moisture storage and vapor diffusion.

The **Moisture storage function** $w(\phi_{\text{w}})$ should be set to characterize the relationship between the amount of water accumulated 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 vapor diffusion properties of building materials:

- **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.

$\textcircled{Q}$	• The Heat Transfer in Building Materials Interface
	• The Heat and Moisture Transport Interfaces
	• Heat and Moisture

📄	See Building Material Library in the *COMSOL Multiphysics Reference Manual* for a description of materials containing the thermal and hygroscopic properties required by the Building Material node.

📄	The **Building Material** node is defined in the spatial frame. The material properties should be entered in the spatial frame, and coupling with a moving frame interface is not supported. See Handling Frames in Heat Transfer for more details.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Specific Media>Building Material
Heat Transfer in Solids>Specific Media>Building Material
Heat Transfer in Fluids>Specific Media>Building Material
Heat Transfer in Porous Media>Specific Media>Building Material
Heat Transfer in Building Materials>Specific Media>Building Material

Ribbon
Physics tab with **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, or **Heat Transfer in Building Materials** selected:

Domains>Specific Media>Building Material

Convectively Enhanced Conductivity

This subnode takes into account convective heat flux by enhancing the fluid thermal conductivity according to the Nusselt number. When added under the **Fluid** or the **Moist Air** node, it sets the fluid velocity to zero, and the user-defined or predefined velocity model input is ignored.

CONVECTIVELY ENHANCED CONDUCTIVITY
The following options are available in the **Nusselt number correlation** list:

- **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. Unfold the **Sketch** section for more details on the required parameters.

- **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. Unfold the **Sketch** section for more details on the required parameters.

- **User defined**, for which a value for Nu should be specified directly.

For the two first options, select **Automatic** (default) or **User defined** to define the **Temperature difference** ΔT. When **Automatic** is selected the temperature difference is

evaluated as the difference between the maximal and the minimal temperature on the exterior boundaries of the feature selection.

DYNAMIC VISCOSITY

This section is not available when the **Convectively Enhanced Conductivity** feature is added under the **Fluid** node with **Fluid type** set as **Moist air**, or under the **Moist Air** node. Else, the **Dynamic viscosity** μ used to calculate the Nusselt number should be set.

📝	When the **Fluid type** is **Moist air**, or when the **Convectively Enhanced Conductivity** feature is added under the **Moist Air** node, the natural convection due to vapor concentration gradients is neglected. Indeed, only thermally induced variations of the density are accounted for in these correlations.

📝	When the **Nonisothermal Flow** multiphysics coupling node is added, the **Convectively Enhanced Conductivity** feature is overridden by the multiphysics coupling node's contribution, and convection is explicitly calculated by using the predefined velocity model input.

▥	*Evaporative Cooling of Water*: Application Library path **Heat_Transfer_Module/Phase_Change/evaporative_cooling**

🔍	[Equivalent Thermal Conductivity Correlations](#)

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Fluids>Fluid>Convectively Enhanced Conductivity
Heat Transfer in Moist Air>Moist Air>Convectively Enhanced Conductivity

More locations are available. For example:

Heat Transfer in Solids>Fluid>Convectively Enhanced Conductivity

Ribbon
Physics tab with **Fluid** or **Moist Air** selected in the model tree:

Attributes>Convectively Enhanced Conductivity

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 take that into account. See Equation 4-172 and Equation 4-173.

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.

⚠	The **Cross Section** feature is disabled when at least one of the multiphysics couplings (Heat Transfer with Surface-to-Surface Radiation, Heat Transfer with Radiation in Participating Media, Heat Transfer with Radiative Beam in Absorbing Media) is active while solving on the same domain. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

🔍	Out-of-Plane Heat Transfer

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Cross Section
Heat Transfer in Solids>Cross Section
Heat Transfer in Fluids>Cross Section
Heat Transfer in Porous Media>Cross Section
Heat Transfer in Building Materials>Cross Section
Bioheat Transfer>Cross Section

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media** or **Bioheat Transfer** selected:

Domains>*interface***>Cross Section**

Fluid

This node uses the following version of the heat equation to model heat transfer in fluids:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \tag{6-5}$$

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

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the fluid density.
- C_p (SI unit: $J/(kg \cdot K)$) is the fluid heat capacity at constant pressure.
- k (SI unit: $W/(m \cdot K)$) 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 more heat sources as separate physics features. See the Heat Source node and the 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 INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include 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 taken from **Common model input**. It corresponds to the `minput.pA` variable, set to 1 atm by default. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section. 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. The last option is **User defined**.

HEAT CONVECTION

The default **Velocity field 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). The **Common model input** option corresponds to the `minput.u` variable. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set values for the **Velocity** components in the **Expression for remaining selection** section.

HEAT CONDUCTION, FLUID

The thermal conductivity k describes the relationship between the heat flux vector **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 **Full** 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 defines 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 between the heat capacity at constant pressure, C_p, and 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 ideal gas law when **Ideal gas** is selected.

When the density is not taken from a **Nonisothermal Flow** or a **Nonisothermal Mixture Model** coupling node, you should select a **Fluid type** option for the specification of the material properties. The available **Fluid type** options are **From material**, **Ideal gas**, and **Gas/Liquid** (default). After selecting a **Fluid type** from the list, further settings display underneath. See Nonisothermal Mixture Model in the *CFD Module User's Guide* for details.

From Material
This option automatically detects whether the material applied on each domain selection is an ideal gas or not, and uses the relevant properties from the **Material** node for either case.

By using the following definition of the density for an ideal gas:

$$\rho = \frac{p_A}{R_s T}$$

with R_s the specific gas constant, p_A the absolute pressure, and T the temperature, the evaluation of the isobaric compressibility coefficient, α_p, and of the isothermal Joule-Thomson coefficient, μ_{JT}, is simplified. This may improve efficiency, when computing pressure work in compressible nonisothermal flows for example, or when modeling inflow conditions.

> 📄 The Air material, from the Built-in materials database, has an **Ideal gas** property group, and is thus detected as an ideal gas by the **Fluid** node. The Liquids and Gases Materials Library, available with some COMSOL products, also provides such materials.

The **Density** and the **Heat capacity at constant pressure** are automatically taken **From material** and no further setting is required.

Automatic calculation of the ratio of specific heats

The **Ratio of specific heats** is calculated automatically using Mayer's relation:

$$\gamma = \frac{\rho C_p \chi_t}{\rho C_p \chi_t - T \alpha_p^2}$$

considering that:

- α_p is the coefficient of thermal expansion (SI unit: 1/K):

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

- χ_t is the isothermal compressibility (SI unit: 1/Pa):

$$\chi_t = -\frac{1}{\rho}\frac{\partial \rho}{\partial p}$$

For the specification of user defined material properties, the Ideal Gas or Gas/Liquid options should be used instead.

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 calculate 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.

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. By default, the **Ratio of specific heats** is set to Automatic, see Automatic calculation of the ratio of specific heats.

Q
- Theory for Heat Transfer in Porous Media
- Theory for Heat Transfer in Fluids

- When the density is taken from a Nonisothermal Flow coupling node, the **Heat capacity at constant pressure** remain to be specified for a general gas or liquid. Select **Ideal gas: ratio of specific heats** or **Ideal gas: heat capacity at constant pressure** to specify only one property in the case of an ideal gas.

- When the Heat Transfer interface is coupled to a Phase Transport interface and to a Fluid Flow interface via the **Nonisothermal Flow** and **Nonisothermal Mixture Model** multiphysics couplings, all the material properties are taken from the coupling nodes. See Nonisothermal Flow and Nonisothermal Mixture Model in the *CFD Module User's Guide* for details.

The Viscous Dissipation (for heat generated by viscous friction), Pressure Work (available with some COMSOL products), and Convectively Enhanced Conductivity subnodes are available 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 https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>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

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Domains>*interface***>Fluid**

Fluid (Porous Medium)

This node defines the velocity field and material properties of the mobile fluid used in the heat transfer equation of the Porous Medium parent node, to model heat transfer in a porous matrix, possibly consisting of several solids, and filled with a mobile fluid, and one or more immobile fluids.

The fluid can be specified as a general gas or liquid, as an ideal gas, or as moist air.

> • The total volume fraction and material properties of immobile fluids can be specified in the Immobile Fluids (Porous Medium) subnode.
>
> • The porosity and material properties of the solid matrix can be specified in the Porous Matrix (Porous Medium, Moist Porous Medium) subnode.
>
> • When the porous medium is a packed bed of pellets, the material properties of the pellets can be specified in the Pellets (Porous Medium) subnode.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

> ⚠️ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button (🗂), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include 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 taken from **Common model input**. It corresponds to the variable minput.pA, set to 1 atm by default. To edit it, click the **Go to Source** button (🗂), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section. 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. The last option is **User defined**.

Concentration

This section can be edited whenever a material property is dependent on concentration, for example, when the **Fluid type** is set to **Moist air** with the **Input quantity** set to **Concentration**.

In the **Concentration** c (SI unit: mol/m^3 or kg/m^3) list, select an existing concentration variable from another physics interface, if there are concentration

variables, **User defined** to enter a value or expression for the concentration, or **Common model input** that corresponds to the minput.c variable.

HEAT CONVECTION

The default **Velocity field 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). The **Common model input** option corresponds to the minput.u variable. To edit it, click the **Go to Source** button (🔲), and in the **Default Model Inputs** node under **Global Definitions**, set values for the **Velocity** components in the **Expression for remaining selection** section.

HEAT CONDUCTION, FLUID

The thermal conductivity k_f describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k_f \nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k_f is taken **From material**. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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.

Note that this section is not available when **Porous medium type** is set to **Local thermal equilibrium** and **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list of the **Porous medium** parent node.

THERMODYNAMICS, FLUID

This section defines the thermodynamics properties of the fluid.

The heat capacity at constant pressure $C_{p,f}$ describes the amount of thermal energy required to produce a unit temperature change in a unit mass.

The ratio of specific heats γ is the ratio between the heat capacity at constant pressure, $C_{p,f}$, and the heat capacity at constant volume, $C_{v,f}$. When using the ideal gas law to describe a fluid, it is sufficient to specify γ to evaluate $C_{p,f}$. 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 ideal gas law when **Ideal gas** is selected.

The ratio of specific heats can be calculated automatically when set to **Automatic** by using Mayer's relation:

$$\gamma = \frac{\rho C_p \chi_t}{\rho C_p \chi_t - T\alpha_p^2}$$

considering that:

- α_p is the coefficient of thermal expansion (SI unit: 1/K):

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

- χ_t is the isothermal compressibility (SI unit: 1/Pa):

$$\chi_t = -\frac{1}{\rho}\frac{\partial \rho}{\partial p}$$

The available options for the **Fluid type** are **Gas/Liquid** (default), **Moist air**, or **Ideal gas**. After selecting a **Fluid type** from the list, other settings are displayed below.

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 necessary 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 calculate the specific gas constant.

- Either the **Heat capacity at constant pressure** $C_{p,f}$ or the **Ratio of specific heats** γ by selecting the option in the **Specify Cp or γ** list. For an ideal gas, it is sufficient to specify either $C_{p,f}$ or the ratio of specific heats, γ, because these properties are interdependent.

Moist Air

If **Moist air** is selected, the thermodynamics properties are defined as a function of the amount of vapor in moist air. The **Input quantity** options available to define this amount are as follows:

- **Vapor mass fraction** to define the ratio between vapor mass and total mass.

- **Concentration** to define the amount of water vapor in the total volume. If this option is selected, a **Concentration** model input is automatically added to the **Model Inputs** section.

- **Moisture content** (default, also called mixing ratio or humidity ratio) to define the ratio between the mass of water vapor and the mass of dry air.

- **Relative humidity** ϕ_w , a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to air saturated with water vapor. The **Relative humidity, temperature condition** T_{ϕ_w} and **Relative humidity, absolute pressure condition** p_{ϕ_w} should be specified.

🔍	Theory for Heat Transfer in Moist Air

📑	When the **Porous Medium type** is set to **Local thermal nonequilibrium** in the Porous Medium parent node, the Initial Values, Heat Source, Thermal Insulation, Symmetry (Heat Transfer Interface), Temperature, Heat Flux, Lumped System Connector, Phase Change Interface, Continuity, Inflow, Outflow, Open Boundary, Boundary Heat Source, Surface-to-Ambient Radiation (Heat Transfer Interface), and Deposited Beam Power features are available under the **Fluid** subnode. These subnodes allow the definition of domain and boundary conditions specific to the fluid phase temperature T_f.

DYNAMIC VISCOSITY

When the **Local thermal nonequilibrium** option is selected in the **Porous medium type** list of the Porous Medium parent node, and **Interstitial convective heat transfer coefficient** is set to **Spherical pellet bed**, the **Dynamic viscosity**, μ, should be set to evaluate the Nusselt number. Note that when **Fluid type** is set to **Moist air**, the moist air viscosity is defined by the node, and this section is not present.

Context Menus
Heat Transfer in Porous Media>Porous Medium>Fluid

Ribbon
Physics tab with **Porous Medium** selected in the model tree:

Attributes>Porous Medium>Fluid

Geothermal Heating

Use this subnode under a Porous Medium node to take into account the contribution of geothermal heat production by adding a source term Q_{geo} to the right-hand side of the heat equation:

$$Q_{\text{geo}} = \rho_{\text{geo}} q_{\text{geo}} f(z_{\text{geo}})$$

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}.

Select an option in the **Distribution type** list for the definition of $f(z_{\text{geo}})$:

- If **Uniform distribution** is selected, $f(z_{\text{geo}}) = 1$ 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 $f(z_{\text{geo}}) = \exp(z_{\text{geo}}/h_{\text{geo}})$, 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 $z_{\text{geo}} = 0$, 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

$$\rho_{geo} = \frac{\displaystyle\sum_i \theta_{si}\rho_{si}}{\displaystyle\sum_i \theta_{si}}$$

- If **Porous media** is selected, it calculates the geothermal density based on all mobile and immobile components of the porous medium:

$$\rho_{geo} = \sum_i \theta_{si}\rho_{si} + \varepsilon_p\rho_f + \theta_{imf}\rho_{imf}$$

- 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
> https://www.comsol.com/products/specifications/

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 rate per unit volume, as a linear heat source, or as a heat rate.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Source** section are defined in the material or spatial frame:

- The default option for the **Heat Source** node is **Solid**, which specifies that the heat source Q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_0 in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

📝 | 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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

HEAT SOURCE

Select an option from the **Heat Source** list:

- For **General source** (default) 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.

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 Battery Design Module, Corrosion Module, Electrodeposition Module, or Fuel Cell &Electrolyzer Module, heat sources from the electrochemical current distribution interfaces are available.

🔍	• Handling Frames in Heat Transfer • Stabilization Techniques in the *COMSOL Multiphysics Reference Manual*

📄	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

| | 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 https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Heat Source
Heat Transfer in Porous Media>Porous Medium>Fluid>Heat Source
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Heat Source

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains>*interface*>Heat Source

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Heat Source

Immobile Fluids (Porous Medium)

This node should be used to model an unsaturated porous matrix whose pore space is not solely filled with liquid and contains one or more gases considered to be immobile fluids; or a porous matrix containing insulated enclosures.

> The **Immobile Fluids** node is available only when **Porous Medium type** is set to **Local thermal equilibrium** in the Porous Medium parent node, with either **Plane layers parallel to heat flow**, **Plane layers perpendicular to heat flow**, **Power law**, or **Equivalent thermal conductivity** option selected in the **Effective thermal conductivity** list.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

IMMOBILE FLUIDS

In this section, the total **Volume fraction** of the immobile fluids, θ_{imf}, should be set.

The total volume fraction of immobile fluids is calculated from

$$\theta_{imf} = \sum_i \theta_{imfi}$$

The volume fraction available for mobile fluids (that is, the effective porosity) is then calculated from

$$\varepsilon_p = 1 - \theta_s - \theta_{imf}$$

where the total volume fraction of immobile solids is calculated from

$$\theta_s = \sum_i \theta_{si}$$

HEAT CONDUCTION, IMMOBILE FLUIDS

The default **Thermal conductivity** k_{imf} use values **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity and other values or expressions in the fields or matrices.

When the thermal conductivity is taken **From material** and more than one **Immobile Fluid** subnode is added under the **Porous Material** node, the effective conductivity for the equivalent immobile fluid is calculated from

$$k_{imf} = \sum_i \theta_{imfi} k_{imfi}$$

In the Porous Medium node, the effective thermal conductivity is defined as:

- **Plane layers parallel to heat flow** (default), which calculates the effective conductivity of the solid-fluid system as the weighted arithmetic mean (or volume average) of the conductivities of the fluids and the porous matrix:

$$k_{eff} = \theta_s k_s + \varepsilon_p k_f + \theta_{imf} k_{imf}$$

- **Plane layers perpendicular to heat flow**, which calculates the effective conductivity of the solid-fluid system as the weighted harmonic mean (or reciprocal average) of the conductivities of the fluids and the porous matrix:

$$\frac{1}{k_{eff}} = \frac{\theta_s}{k_s} + \frac{\varepsilon_p}{k_f} + \frac{\theta_{imf}}{k_{imf}}$$

- **Power law**, which calculates the effective conductivity of the solid-fluid system as the weighted geometric mean of the conductivities of the fluids and the porous matrix:

$$k_{eff} = k_s^{\theta_s} \cdot k_f^{\varepsilon_p} \cdot k_{imf}^{\theta_{imf}}$$

THERMODYNAMICS, IMMOBILE FLUIDS

The specific heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass of the immobile fluid.

The **Density** ρ_{imf} and the **Specific heat capacity** $C_{p,\,imf}$ should be specified. When the density and specific heat capacity are taken **From material** and more than one **Immobile Fluid** subnode is added under the **Porous Material** node, the effective properties are obtained by applying a volume average model:

$$\rho_{imf} = \sum_i \theta_{imfi}\rho_{imfi}$$

$$C_{p,\,imf} = \frac{\sum_i \theta_{imfi}\rho_{imfi}C_{p,\,imfi}}{\sum_i \theta_{imfi}\rho_{imfi}}$$

In the Porous Medium node, the effective volumetric heat capacity at constant pressure is defined as:

$$(\rho C_p)_{eff} = \theta_s\rho_s C_{p,\,s} + \varepsilon_p\rho_f C_{p,\,f} + \theta_{imf}\rho_{imf}C_{p,\,imf}$$

> The **Immobile Fluids** node requires the Subsurface Flow Module. For a detailed overview of the functionality available in each product, visit
> https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Porous Media>Porous Medium>Immobile Fluids

Ribbon
Physics tab with **Porous Medium** selected in the model tree:

Attributes>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.

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 an **Ambient Properties** node under **Definitions**.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Initial Values
Heat Transfer in Porous Media>Porous Medium>Fluid>Initial Values
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Initial Values

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains>*interface*>Initial Values

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Initial Values

Initial Values (Radiative Beam in Absorbing Medium Interface)

This node adds an initial value for the radiative intensity that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter a value or expression for the initial value of the **Radiative intensity** I (SI unit: W/ m^2). This initial value is used by all the beams defined in the model.

LOCATION IN USER INTERFACE

Context Menus
Radiative Beam in Absorbing Media>Initial Values

Ribbon
Physics tab with **Radiative Beam in Absorbing Media** selected:

Domains>Initial Values

Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node adds an initial value for the radiative intensity I (when **Discrete ordinates method** is selected as the **Radiation discretization method**) or the incident radiation G (when **P1 approximation** is selected as the **Radiation discretization method**) that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Discrete Ordinates Method

When **Discrete ordinates method** is selected as the **Radiation discretization method** for the physics interface, you should define the initial radiative intensity using one of the following options, depending on the **Wavelength dependence of radiative properties** set in the **Participating Media Settings** section of the interface.

If the **Wavelength dependence of radiative properties** is **Constant**, choose between the following options:

- Use the **Blackbody/Graybody** assumption and set a value for the **Initial temperature** T, to define $I=I_b(T)$. This option is only available in the Radiation in Participating Media interface.

- Set a **User defined** value for the **Initial radiative intensity** I. The default is 0 in the Radiation in Absorbing-Scattering Media interface, and `rpm.Ibinit` in the Radiation in Participating Media interface.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, choose among the following options:

- When **Initial value** is set to **Blackbody/Graybody**, enter a value for the **Initial temperature** T, to define $I=I_b(T)$. This option is only available in the Radiation in Participating Media interface.

- When **Initial value** is set to **User defined for each band**, enter a value for the **Initial radiative intensity** for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

- When **Initial value** is set to **User defined distribution**, enter a value or expression for the **Initial radiative intensity distribution** I_λ. The wavelength may be accessed through the `rpm.lambda` or `rasm.lambda` variable. This distribution is integrated on each spectral band to obtain the radiative intensity $I_{i,k}$ for each spectral band.

P1 Approximation

When **P1 approximation** is selected as the **Radiation discretization method** for the physics interface, you should define the initial incident radiation using one of the following options, depending on the **Wavelength dependence of radiative properties** set in the **Participating Media Settings** section of the interface.

If the **Wavelength dependence of radiative properties** is **Constant**, choose between the following options:

- Use the **Blackbody/Graybody** assumption and set a value for the **Initial temperature** T, to define $G = 4\pi I_{\mathrm{b}}(T)$. This option is only available in the Radiation in Participating Media interface.

- Set a **User defined** value for the **Initial incident radiation** G. The default is 0 in the Radiation in Absorbing-Scattering Media interface, and `(4*pi)*rpm.Ibinit` in the Radiation in Participating Media interface.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, choose among the following options:

- When **Initial value** is set to **Blackbody/Graybody**, enter a value for the **Initial temperature** T, to define $G = 4\pi I_{\mathrm{b}}(T)$. This option is only available in the Radiation in Participating Media interface.

- When **Initial value** is set to **User defined for each band**, enter a value for the **Initial incident radiation** for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

- When **Initial value** is set to **User defined distribution**, enter a value or expression for the **Initial incident radiation distribution** G_λ. The wavelength may be accessed through the `rpm.lambda` or `rasm.lambda` variable. This distribution is integrated on each spectral band to obtain the incident radiation G_k for each spectral band.

🔍   Theory for Radiation in Participating Media

LOCATION IN USER INTERFACE

Context Menus
Radiation in Participating Media>Initial Values
Radiation in Absorbing-Scattering Media>Initial Values

Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Domains>Initial Values

Irreversible Transformation

This subnode should be used to model thermally induced irreversible transformations in solids.

IRREVERSIBLE TRANSFORMATION

Three models are available to define the material transformation. Select the **Transformation model** — **Temperature threshold** (default), **Arrhenius kinetics**, or **User defined**. The first two models use integral forms over time to express the fraction of transformation θ_{it} as a function of temperature, while you can set it manually with the third option.

Temperature Threshold

For **Temperature threshold**, select the type of analysis — **Overheating analysis** (default) or **Overcooling analysis**, depending on the expected temperature variations. See the Parameters section for the additional settings specific to each type of analysis.

Arrhenius Kinetics

For **Arrhenius kinetics**, define the parameters used in the Arrhenius equation to calculate the degree of transformation (see Arrhenius Kinetics for more details):

- **Frequency factor** A in the Arrhenius equation. Default is taken **From material**. For **User defined** enter a value or an expression.

- **Activation energy** ΔE in the Arrhenius equation. Default is taken **From material**. For **User defined** enter a value or an expression.

- **Polynomial order** n of the $(1-\alpha)$ factor to define a polynomial Arrhenius kinetics equation.

- **Enthalpy change** L to define the enthalpy variation associated with the transformation. The following heat source is added to the right-hand side of Equation 6-15 in Solid node:

$$Q = -\rho L \frac{\partial \theta_{it}}{\partial t}$$

A user defined value can be set for the initial irreversible transformation indicator in the Initial Values section.

User Defined
Enter values or expressions for the **Enthalpy change** L and the **Fraction of transformation** θ_{it} to define the heat source associated with the transformation as:

$$Q = -\rho L \frac{\partial \theta_{it}}{\partial t}$$

Specify Different Material Properties
Specific thermodynamics properties before and after complete transformation may be defined by selecting the **Specify different material properties for the transformed state** check box.

Choose a **Transformed material**, which can point to any material in the model. The default uses the **Domain material**. The properties before transformation are the ones specified in the **Heat Conduction, Solid** and **Thermodynamics, Solid** sections of the parent Solid node. The effective material properties are dynamically updated with the transformation evolution.

PARAMETERS

Overheating Analysis
Enter values for:

- **Transformation temperature** $T_{it, h}$ to define the (high) temperature that the solid needs to reach to start getting transformed.
- **Transformation time** $t_{it, h}$ to define the time needed for the complete transformation to happen while the temperature is above $T_{it, h}$.
- **Enthalpy change** $L_{it, h}$ to define the enthalpy variation associated with transformation due to overheating. The following heat source is added to the right-hand side of Equation 6-15 in Solid node:

$$Q = -\rho L_{it, h} \frac{\partial \theta_{it}}{\partial t}(T > T_{it, h})$$

Overcooling Analysis
Enter values for:

- **Transformation temperature** $T_{it, c}$ to define the (low) temperature that the solid needs to reach to start getting transformed.

- **Transformation time** $t_{it, c}$ to define the time needed for the complete transformation to happen while the temperature is below $T_{it, c}$.

- **Enthalpy change** $L_{it, c}$ to define the enthalpy variation associated with transformation due to overcooling. The following heat source is added to the right-hand side of Equation 6-15 in Solid node:

$$Q = \rho L_{it, c} \frac{\partial \theta_{it}}{\partial t} (T < T_{it, c})$$

A user defined value can be set for the initial irreversible transformation indicator in the Initial Values section.

HEAT CONDUCTION

This section is available when the **Specify different material properties for the transformed state** check box is selected.

Select a **Thermal conductivity** k_d — **From material** (default) or **User defined**, to be used for transformed solid. For **User defined** choose **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity and enter another value or expression in the field or matrix.

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics physics interface for example), you can select on which frame the **Thermal conductivity** k_d is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables `ht.k_dxx`, `ht.k_dyy`, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables `ht.k_dxx`, `ht.k_dyy`, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{d(x, y, z)} = \mathbf{R} k_{d(X, Y, Z)} \mathbf{R}^{\mathbf{T}}$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

This section is available when the **Specify different material properties for the transformed state** check box is selected.

Select a **Density** ρ_d and **Heat capacity at constant pressure** $C_{p,\,d}$ — **From material** (default) or **User defined**, to be used for transformed solid. The heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass.

INITIAL VALUES

This section is available when the **Transformation model** is set to **Temperature threshold** or **Arrhenius kinetics**. Set a value or expression for the **Initial irreversible transformation indicator**, α_{init}, to be used as an initial condition for any of the time integral analyses.

> For a detailed overview of the functionality available in each product, visit
> https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids>Solid>Irreversible Transformation

More locations are available. For example:

Heat Transfer in Fluids>Solid>Irreversible Transformation

Ribbon
Physics tab with **Solid** selected in the model tree:

Attributes>Irreversible Transformation

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.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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** (![icon]). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

ISOTHERMAL DOMAIN

Two options are available for the **Temperature definition** into the **Isothermal Domain**:

- If **From heat balance** (default) is selected, the temperature is calculated as the solution of a reduced form of the heat equation due to spatial homogeneity. See Equation 4-54.

- If **From prescribed temperature** is selected, the temperature is set to the domain **Temperature** T_0 that needs to be specified. For **User defined**, enter a value or expression for the **Temperature** T_0. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

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.

![icon]	It is not possible to couple an **Isothermal Domain** with the **Laminar Flow** interface through the **Nonisothermal Flow** multiphysics condition.

![icon]	- 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.

	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 in Solids and Fluids>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

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Domains>*interface*>Isothermal Domain

Liquid Water (Moist Porous Medium)

This subnode defines the velocity field $\mathbf{u}_l$ and the saturation s_l of the liquid water phase used in Equation 6-6 of the Moist Porous Medium parent node. The density ρ_l (SI unit: kg/m^3), the heat capacity at constant pressure $C_{p,l}$ (SI unit: $J/(kg \cdot K)$), and the thermal conductivity k_l (SI unit: $W/(m \cdot K)$) are defined as functions of temperature.

LIQUID WATER PROPERTIES

The **Liquid water saturation**, s_l, used to evaluate the effective thermodynamics properties in the Moist Porous Medium parent node, should be set.

The default **Velocity field, liquid water**, $\mathbf{u}_l$, 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).

If the node was added automatically after selecting the predefined multiphysics interface **Heat and Moisture Transport**, the liquid saturation and liquid velocity of the

multiphysics node **Heat and Moisture** are used by default, and the section is not editable. To edit the **Liquid water saturation** and **Velocity field, liquid water** fields, click **Make All Model Inputs Editable** (![icon]).

Q
- [Theory for Heat Transfer in Moist Porous Media](#)
- [Heat and Moisture](#)

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Moist Porous Media>Moist Porous Medium>Liquid Water

Ribbon
Physics tab with **Moist Porous Medium** selected in the model tree:

Attributes>Liquid Water

Moist Air (Heat Transfer Interface)

This node should be used to model heat transfer by convection and diffusion in moist air, using moisture-dependent thermodynamics properties. It solves [Equation 6-5](#), with density ρ (SI unit: kg/m^3), heat capacity at constant pressure C_p (SI unit: J/(kg·K)), and thermal conductivity k (SI unit: W/(m·K)) defined as a function of the amount of vapor in moist air.

MODEL INPUT
This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature
This section is available when material properties are temperature-dependent. 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** (![icon]). The available options are **User defined** (default), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include enthalpy (the energy flux, for example).

The default **Absolute pressure** p_A is taken from **Common model input**. It corresponds to the `minput.pA` variable, set to 1 atm by default. To edit it, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section. 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. The last option is **User defined**.

Concentration

This section can be edited whenever a material property is dependent on concentration, for example, when the **Fluid type** is set to **Moist air** with the **Input quantity** set to **Concentration**.

In the **Concentration** c (SI unit: mol/m^3 or kg/m^3) list, select an existing concentration variable from another physics interface, if there are concentration variables, **User defined** to enter a value or expression for the concentration, or **Common model input** that corresponds to the `minput.c` variable.

HEAT CONVECTION

The default **Velocity field 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). The **Common model input** option corresponds to the `minput.u` variable. To edit it, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set values for the **Velocity** components in the **Expression for remaining selection** section.

THERMODYNAMICS, MOIST AIR

The **Input quantity** options available to define the amount of vapor in moist air are as follows:

- **Vapor mass fraction** ω to define the ratio between vapor mass and total mass.

- **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** x_{vap}, also called mixing ratio or humidity ratio, to define the ratio between water vapor mass and dry air mass.

- **Relative humidity** ϕ_{w} (default), a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to air saturated with water vapor. The **Relative humidity, temperature condition** $T_{\phi_{\mathrm{w}}}$ and **Relative humidity, absolute pressure condition** $p_{\phi_{\mathrm{w}}}$ should be specified.

Depending on the selected **Input quantity**, enter values or expressions for the **User defined** option, select an ambient value (for example, an **Ambient relative humidity** defined in an **Ambient Properties** node under **Definitions**), or select a value from a Moisture Transport interface.

Q	Theory for Heat Transfer in Moist Air

📄	With some COMSOL products, the Viscous Dissipation (for heat generated by viscous friction), Pressure Work, and Convectively Enhanced Conductivity subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

▥	*Evaporative Cooling of Water*: Application Library path **Heat_Transfer_Module/Phase_Change/evaporative_cooling**

Q	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Specific Media>Moist Air

Heat Transfer in Solids>Specific Media>Moist Air

Heat Transfer in Fluids>Specific Media>Moist Air

Heat Transfer in Porous Media>Specific Media>Moist Air

Heat Transfer in Building Materials>Specific Media>Moist Air

Bioheat Transfer>Specific Media>Moist Air

Physics tab with **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Domains>Specific Media>Moist Air

Moist Air (Moist Porous Medium)

This subnode defines the velocity field $\mathbf{u}_g$ and the moisture-dependent thermodynamics properties used for the moist air phase in Equation 6-6 of the Moist Porous Medium parent node. The density ρ_g (SI unit: kg/m^3), the heat capacity at constant pressure $C_{p,g}$ (SI unit: J/(kg·K)), and the thermal conductivity k_g (SI unit: W/(m·K)) are defined as functions of the amount of vapor in the moist air phase.

See Heat Convection and Thermodynamics, Moist Air for details on settings.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Moist Porous Media>Moist Porous Medium>Moist Air

Ribbon

Physics tab with **Moist Porous Medium** selected in the model tree:

Attributes>Moist Air

Moist Porous Medium

This node uses the following version of the heat equation to model heat transfer in a porous matrix, possibly consisting of several solids, and filled with moist air and liquid water:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + (\rho_g C_{p,g} \mathbf{u}_g + \rho_l C_{p,l} \mathbf{u}_l) \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + Q_{\text{evap}} \tag{6-6}$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \tag{6-7}$$

See Theory for Heat Transfer in Moist Porous Media for more details on the variables in Equation 6-6 and Equation 6-7.

The effective volumetric heat capacity at constant pressure is defined as

$$(\rho C_p)_{\text{eff}} = \theta_s \rho_s C_{p,\,s} + \varepsilon_p[(1 - s_1)\rho_g C_{p,\,g} + s_1 \rho_1 C_{p,\,1}]$$

where s_1 is the liquid water saturation.

If the porous matrix consists of several solids i of volume fraction θ_{si}, heat capacity $C_{p,si}$, and density ρ_{si}, the above equation is modified as follows:

$$(\rho C_p)_{\text{eff}} = \sum_i \theta_{si} \rho_{si} C_{p,\,si} + \varepsilon_p[(1 - s_1)\rho_g C_{p,\,g} + s_1 \rho_1 C_{p,\,1}]$$

Note that the velocity fields in moist air and liquid water, $\mathbf{u}_g$ and $\mathbf{u}_l$, can be defined either as an analytic expression or as the velocity field from a Fluid Flow interface. They should be interpreted as Darcy velocities, that is, asthe volume flow rate per unit cross sectional area.

Q (SI unit: W/m^3) can be any source (or sink) of heat. Add one or more heat sources as separate physics features. See Heat Source node for example. When the Heat and Moisture multiphysics coupling is active, this term includes the diffusive flux of thermal enthalpy and the liquid capillary flux, calculated from the moisture transport equation.

Q_{evap} (SI unit: W/m^3) is the source (or sink) of heat due to phase change of water. When the Heat and Moisture multiphysics coupling is active, it calculates this term from the moisture transport equation.

COORDINATE SYSTEM SELECTION

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate system**, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The subnodes of **Moist Porous Medium** inherit these coordinate system settings. In particular, the **Velocity field** (in Moist Air (Moist Porous Medium) subnode) and the **Dry bulk thermal conductivity** and **Solid phase thermal conductivity** (in Porous Matrix (Porous Medium, Moist Porous Medium) subnode) should be set according to the coordinate system selected in this section.

See Coordinate Systems in the *COMSOL Multiphysics Reference Manual* for more details.

POROUS MEDIUM MODEL SETTINGS

This section defines the averaging model for the computation of the **Effective thermal conductivity** k_{eff}, taking into account the properties of the solid matrix, moist air, and liquid water. The following options are available:

- **Plane layers parallel to heat flow** (default), which calculates the effective conductivity of the moist porous medium as the weighted arithmetic mean (or volume average) of the conductivities of the porous matrix, moist air, and liquid water:

$$k_{eff} = \theta_s k_s + \varepsilon_p [(1 - s_l) k_g + s_l k_l]$$

- **Plane layers perpendicular to heat flow**, which calculates the effective conductivity of the moist porous medium as the weighted harmonic mean (or reciprocal average) of the conductivities of the porous matrix, moist air, and liquid water:

$$\frac{1}{k_{eff}} = \frac{\theta_s}{k_s} + \frac{\varepsilon_p}{(1 - s_l) k_g + s_l k_l}$$

- **Power law**, which calculates the effective conductivity of the moist porous medium as the weighted geometric mean of the conductivities of the porous matrix, moist air, and liquid water:

$$k_{eff} = k_s^{\theta_s} \cdot [(1 - s_l) k_g + s_l k_l]^{\varepsilon_p}$$

If the porous matrix consists of several solids i of volume fraction θ_{si} and thermal conductivity k_{si}, the above equations are modified as follows:

- **Plane layers parallel to heat flow**:

$$k_{eff} = \sum_i \theta_{si} k_{si} + \varepsilon_p [(1 - s_l) k_g + s_l k_l]$$

- **Plane layers perpendicular to heat flow**:

$$\frac{1}{k_{eff}} = \sum_i \theta_{si} \frac{\sum_i \theta_{si}}{\sum_i \theta_{si} k_{si}} + \frac{\varepsilon_p}{(1 - s_l) k_g + s_l k_l}$$

- **Power law**:

$$k_{\text{eff}} = \left(\frac{\sum_i \theta_{si} k_{si}}{\sum_i \theta_{si}} \right)^{\sum_i \theta_{si}} \cdot \left[(1 - s_1) k_g + s_1 k_1 \right]^{\varepsilon_p}$$

The velocity field and the moisture content of moist air can be specified in the Moist Air (Moist Porous Medium) subnode.

The liquid water saturation and velocity field can be specified in the Liquid Water (Moist Porous Medium) subnode.

The porosity and material properties of the solid matrix can be specified in the Porous Matrix (Porous Medium, Moist Porous Medium) subnode.

     See Porous Material in the *COMSOL Multiphysics Reference Manual*.

- Theory for Heat Transfer in Moist Porous Media
- Theory for Heat Transfer in Moist Air
- Heat and Moisture

With some COMSOL products, the Thermal Dispersion, Viscous Dissipation, and Geothermal Heating subnodes are available 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 https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Moist Porous Media>Specific Media>Moist Porous Medium

Ribbon

Physics tab with **Heat Transfer in Moist Porous Media** selected:

Domains>Specific Media>Moist Porous Medium

Opacity (Surface-to-Surface Radiation Interface)

This node 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 **Emitted radiation direction** is defined by **Opacity controlled** in surface-to-surface boundary features (Diffuse Mirror (Surface-to-Surface Radiation Interface), Diffuse Surface (Surface-to-Surface Radiation Interface), Prescribed Radiosity (Surface-to-Surface Radiation Interface), Opaque Surface (Surface-to-Surface Radiation Interface)), surface-to-surface radiation propagates in non-opaque domains. Alternatively the **Emitted radiation direction** can be defined using the normal orientation or on both sides of boundaries. In this case the **Opacity** node is ignored. Note that on boundaries where the Semitransparent Surface (Surface-to-Surface Radiation Interface) is applied, the **Emitted radiation direction** is **Both sides**, and the **Opacity** node is ignored.

By default, all the domains and the exterior of the Surface-to-Surface Radiation interface are considered as transparent. To set the exterior as opaque, change the **Selection** of the **Opacity** node to **All voids** in the **Domain Selection** section, and set it as **Opaque**. Depending on the geometry, an infinite void and several finite voids may be available for the selection. When the Heat Transfer with Surface-to-Surface Radiation multiphysics coupling is active, it controls the default opacity.

OPACITY

Depending on the **Wavelength dependence of radiative properties** defined in the Radiation Settings section of the physics interface settings, different sections display underneath:

- When **Wavelength dependence of radiative properties** is set to **Constant**, select **Opaque** or **Transparent** to set the domain's opacity type.

- When **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, set the opacity for each spectral band by selecting the

corresponding check box in the **Opaque** column of the table. By default the check box is selected for all spectral bands.

IIII	*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
Surface-to-Surface Radiation>Opacity

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected in the model tree:

Domains>Opacity

Optically Thick Participating Medium

This node should be used when radiation occurs in a medium with high optical thickness. It models the heating due to the propagation of the rays by modifying the thermal conductivity with

$$k_R = \frac{16 n_r^2 \sigma T^3}{3 \beta_R}$$

where n_r is the refractive index (dimensionless), σ is the Stefan-Boltzmann constant (SI unit: $W/(m^2 \cdot K^4)$), and β_R is the extinction coefficient.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature
This section is available when material properties are temperature-dependent. 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** (![icon]). The available options are **User defined** (default), **Common model input** (the minput.T variable, set to

293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button (⬚), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

OPTICALLY THICK PARTICIPATING MEDIUM

This section defines the method of computation of the properties of the participating medium.

First, define the **Refractive index** n_r (dimensionless) of the participating medium.The default value, 1, is usually a good approximation for the air refractive index.

Then, 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 $\beta_R = \kappa + \sigma_s$ 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 defines the absorption property of the participating medium. It is available when **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 defines the scattering property of the participating medium, and is available when **Absorption and scattering coefficients** is selected from the **Specify media properties** list. The **Scattering coefficient** σ_s should be specified.

🔍	See Participating Medium (Radiation in Participating Media Interface) for the modeling of linear or polynomial anisotropic scattering.

▥	*Radiative Cooling of a Glass Plate*: Application Library path **Heat_Transfer_Module/Thermal_Radiation/glass_plate**

Context Menus

Heat Transfer in Solids>Solid>Optically Thick Participating Medium

Heat Transfer in Solids>Fluid>Optically Thick Participating Medium

Heat Transfer in Solids>Porous Medium>Optically Thick Participating Medium

Heat Transfer in Solids>Biological Tissue>Optically Thick Participating Medium

Heat Transfer in Solids>Building Material>Optically Thick Participating Medium

Ribbon

Physics tab with **Solid**, **Fluid**, **Porous Medium**, **Biological Tissue**, or **Building Material** selected in the model tree:

Attributes>Optically Thick Participating Medium

Out-of-Plane Heat Flux

The 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 6-8 and Equation 6-9:

$$d_z \rho C_p \frac{\partial T}{\partial t} - \nabla \cdot \mathbf{q} = d_z Q + q_0 \tag{6-8}$$

$$d_z \rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) = \nabla \cdot \mathbf{q} + d_z Q + q_0 \tag{6-9}$$

$$q_0 = q_{0, \mathrm{u}} + q_{0, \mathrm{d}}$$

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 in the **Flux type** list are **General inward heat flux**, **Convective heat flux**, and **Nucleate boiling heat flux**. The settings are the same as for the Heat Flux node.

DOWNSIDE INWARD HEAT FLUX

The available options in the **Flux type** list are **General inward heat flux**, **Convective heat flux**, and **Nucleate boiling heat flux**. The settings are the same as for the Heat Flux node.

> ❗ The **Out-of-Plane Heat Flux** feature is disabled when at least one of the multiphysics couplings (Heat Transfer with Surface-to-Surface Radiation, Heat Transfer with Radiation in Participating Media, Heat Transfer with Radiative Beam in Absorbing Media) is active while solving on the same domain. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

> 🔍 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 in Solids and Fluids>Out-of-Plane>Out-of-Plane Heat Flux

Heat Transfer in Porous Media>Porous Medium>Fluid>Out-of-Plane Heat Flux

Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Out-of-Plane Heat Flux

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains>*interface***>Out-of-Plane Heat Flux**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>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 6-8 or Equation 6-9:

$$\varepsilon_u \sigma (T_{amb, u}^4 - T^4) + \varepsilon_d \sigma (T_{amb, d}^4 - T^4)$$

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** $T_{amb, u}$. The default value is approximately room temperature, 293.15 K (20°C). Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

Follow the instructions for the **Upside Parameters** section to define the downside parameters ε_d and $T_{amb,\, d}$.

⚠	The **Out-of-Plane Radiation** feature is disabled when at least one of the multiphysics couplings (Heat Transfer with Surface-to-Surface Radiation, Heat Transfer with Radiation in Participating Media, Heat Transfer with Radiative Beam in Absorbing Media) is active while solving on the same domain. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

🔍	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 in Solids and Fluids>Out-of-Plane>Out-of-Plane Radiation
Heat Transfer in Porous Media>Porous Medium>Fluid>Out-of-Plane Radiation
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Out-of-Plane Radiation

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains>*interface***>Out-of-Plane Radiation**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Out-of-Plane Radiation

Participating Medium (Radiation in Participating Media 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 calculates the heating due to the propagation of the rays, and takes into account the absorbed, the emitted, and the scattered radiation, depending on the **Radiation discretization method** selected in the Participating Media Settings section of the interface.

It calculates the radiative heat source term Q (SI unit: W/m^3), defined by:

$$Q = \kappa(G - 4\pi I_b(T))$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2).
- $I_b(T)$ 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)).
- T is the temperature (SI unit: K)

G is defined by

$$G = \int_{4\pi} I(\Omega) d\Omega$$

where

- $I(\Omega)$ is the radiative intensity (SI unit: W/(m^2·sr)) at a given position following the Ω direction, that satisfies the *radiative transfer equation*

$$\Omega \cdot \nabla I(\Omega) = \kappa I_{\mathrm{b}}(T) - \beta I(\Omega) + \frac{\sigma_{\mathrm{s}}}{4\pi} \int_0^{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega'$$

- $\beta = \kappa + \sigma_{\mathrm{s}}$ 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)

If the **Radiation discretization method** is **Discrete ordinates method**, G is defined by

$$G = \sum_{i=1}^{N} \omega_i I_i \qquad (6\text{-}10)$$

and

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_{\mathrm{b}}(T) - \beta I_i + \frac{\sigma_{\mathrm{s}}}{4\pi} \sum_{j=1}^{N} \omega_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \qquad (6\text{-}11)$$

where

- $\mathbf{S}_i$ is the ith discrete ordinate.
- I_i is the ith component of the radiative intensity.
- ω_j is the ith quadrature weight.

If the **Radiation discretization method** is **P1 approximation**, G is the solution of the following equation

$$\nabla \cdot (D_{\mathrm{P1}} \nabla G) - \kappa(G - 4\pi I_{\mathrm{b}}) = 0 \qquad (6\text{-}12)$$

where D_{P1} is the P1 diffusion coefficient.

When **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands** in the **Participating Media Settings** section of the interface, Equation 6-10 and Equation 6-11 are solved for each spectral band k: $I_{i,k}$ is the ith component of the radiative intensity for spectral band k, and G_k is the incident radiation for spectral band k.

The characteristics of the two available radiation discretization methods are summarized in the following table.

TABLE 6-3: DISCRETIZATION METHODS FOR PARTICIPATING MEDIUM (RPM INTERFACE)

OPTION	DOM	P1
Optical thickness validity	All	$\tau >> 1$
Absorption modeling	Yes	Yes
Emission modeling	Yes	Yes
Scattering modeling	Linear Polynomial	Linear
Computational cost	High: up to 80 additional degrees of freedom (I_i)	Medium: 1 additional degree of freedom (G)

If radiative emission may be neglected, the Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface) node may be used instead. The table below describes the different effects accounted for by the interfaces found under the **Heat Transfer>Radiation** branch ( ).

TABLE 6-4: RADIATION EFFECTS CALCULATED BY THE RADIATION INTERFACES

RADIATION EFFECT	RPM	RASM	RBAM
Absorption	Yes	Yes	Yes
Scattering	Yes	Yes	No
Emission	Yes	No	No

MODEL INPUT

This section contains fields and values that are inputs for expressions defining 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. The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T`

variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

FRACTIONAL EMISSIVE POWER

This section is available when the **Wavelength dependence of radiative properties** is defined as **Solar and ambient** or **Multiple spectral bands** for the physics interface (see Participating Media Settings).

When the **Fractional emissive power** is **Blackbody/Graybody**, the fractional emissive power is automatically calculated for each spectral band as a function of the band endpoints and temperature.

When the **Fractional emissive power** is **User defined on each band**, define the **Fractional emissive power**, FEP_k 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.

ABSORPTION

The **Absorption coefficient** κ should be specified. It defines the amount of radiation, $\kappa I(\Omega)$, that is absorbed by the medium.

The **Absorption coefficient** κ (SI unit: $1/m$) uses values **From material** by default.

For **User defined**, set a value or expression. You can define a temperature-dependent absorption coefficient using the variable rasm.T.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength may be accessed through the rasm.lambda variable. Any expression set for the absorption coefficient is then averaged on each spectral band to obtain a piecewise constant absorption coefficient. If the average value of the absorption coefficient on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

When **Absorption coefficient** is set to **User defined for each band**, enter a value for the **Absorption coefficient** for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

SCATTERING

This section defines the scattering property of the participating medium. The **Scattering coefficient** σ_s should be specified.

The **Scattering coefficient** σ_s (SI unit: $1/m$) uses values **From material** by default.

For **User defined**, set a value or expression. You can define a temperature-dependent scattering coefficient using the variable `rasm.T`.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength may be accessed through the `rasm.lambda` variable. Any expression set for the scattering coefficient is then averaged on each spectral band to obtain a piecewise constant scattering coefficient. If the average value of the scattering coefficient on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

When **Scattering coefficient** is set to **User defined for each band**, enter a value for the **Scattering coefficient** for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

Choose in addition the **Scattering type**: **Isotropic**, **Linear anisotropic**, **Polynomial anisotropic** (only with **Discrete ordinates method**), or **Henyey-Greenstein** (only with **Discrete ordinates method**).

This setting provides options to approximate the scattering phase function ϕ using the cosine of the scattering angle, μ_0:

- **Isotropic** (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.

- For **Henyey-Greenstein** it defines the scattering phase function as

$$\phi(\mu_0) = \frac{1}{K} \cdot \frac{1 - \eta^2}{(1 + \eta^2 - 2\eta\mu_0)^{3/2}}$$

where $-1 < \eta < 1$ is the anisotropy parameter and K is defined as follows to produce a normalized phase function:

$$K = \frac{1}{4\pi} \cdot \int\limits_{4\pi} \frac{1 - \eta^2}{(1 + \eta^2 - 2\eta\mu_0)^{3/2}} d\Omega$$

For **Linear anisotropic** and **Polynomial anisotropic**, select the **Normalize phase function** check box to define a phase function such as

$$\int\limits_{4\pi} \phi(\mu_0) d\Omega = 4\pi$$

The normalization is automatically applied for the **Henyey-Greenstein** option.

For **Linear anisotropic**, **Polynomial anisotropic**, and **Henyey-Greenstein**, select the **Wavelength-dependent scattering type** check box to average each **Legendre coefficient** $a_1, ..., a_{12}$ or the **Anisotropy parameter** μ to obtain piecewise constant coefficients on each spectral band.

> • 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 Cooling of a Glass Plate: Application Library path **Heat_Transfer_Module/Thermal_Radiation/glass_plate**

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>Participating Medium

Physics tab with **Radiation in Participating Media** selected:

Domains>Radiation in Participating Media>Participating Medium

Pellets (Porous Medium)

This node defines the porosity and material properties of the pellets contained in a packed bed, used in the heat transfer equations of the Porous Medium parent node, when the **Packed bed** option is selected in the **Porous medium type** list. The coupling condition between the pellets and the fluid phase of the packed bed can be specified in the Pellet-Fluid Interface (Porous Medium) subnode.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button ($\boxed{=_{\uparrow}}$), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

Other options are **User defined** and all the temperature variables from the physics interfaces included in the model.

> ⚠ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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** (![icon]). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to

293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (📝), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

POROSITIES

The default **Bed porosity** ε_p of the packed bed is taken **From material**. In a **Porous Material** node, the porosity is defined as follows:

$$\varepsilon_p = 1 - \sum_i \theta_{si} - \sum_i \theta_{pe} - \sum_i \theta_{imfi}$$

where θ_{si}, θ_{pe} and θ_{imfi} are the volume fractions of the **Solid**, **Pellet**, and **Immobile Fluid** subnodes under the **Porous Material** node.

For **User defined**, enter a value or expression.

As porosity can be considered within the pellets, set the **Pellet porosity**, ε_{pe}. By default it is taken **From material**. Go to the **Pellet** subnode of the **Porous Material** node to enter a value or expression.

HEAT CONDUCTION, PELLETS

Set the thermal conductivity of the porous matrix within the pellets, k_{pe}. It is used to compute the effective thermal conductivity of the pellet filled with fluid, $k_{pe,eff}$, defined by

$$k_{pe,\,eff} = (1 - \varepsilon_{pe})k_{pe} + \varepsilon_{pe}k_f$$

The default **Thermal conductivity** k_{pe} is taken **From material**. For **User defined**, enter a value or expression. A scalar value should be set as it is used in the 1D microscale equation.

THERMODYNAMICS, PELLETS

This section defines the thermodynamics properties of the porous matrix within the pellets.

The specific heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit of mass of the pellets.

The **Density** ρ_{pe} and the **Specific heat capacity** $C_{p,\,pe}$ should be specified. For **From Material** option, see Material Density in Features Defined in the Material Frame if a temperature-dependent density should be set.

These properties are used to compute the effective volumetric heat capacity at constant pressure inside the pellet filled with fluid, $(\rho C_p)_{\text{pe,eff}}$, defined by

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

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Porous Media>Porous Medium>Pellets

Ribbon
Physics tab with **Porous Medium** selected in the model tree:

Attributes>Pellets

Pellet-Fluid Interface (Porous Medium)

This node defines the coupling condition between the fluid phase and the pellets, when the **Packed bed** option is selected in the **Porous medium type** list of the Porous Medium parent node.

PELLET-FLUID INTERFACE

Select an option in the **Coupling condition** list to couple the macroscale heat equation solved in the fluid phase and the microscale equation solved in the pellets to account for radial thermal conduction.

With the **Continuous temperature** option (default), the temperature at the outer surface of the pellets and the fluid temperature are enforced to be equal, and the resulting heat flux is applied in the fluid and pellets equations. See the Constraint Settings section below for advanced settings relative to the continuity constraint.

With the **Convective heat flux** option, a heat transfer term, proportional to the temperature difference between the fluid phase and the outer surface of the pellets, is applied in the equations of both phases. The heat transfer term is applied in an averaged way in the fluid equation, by multiplying it by the **Specific surface area**, S_{b}. Either use the **Automatic** expression for S_{b}, defined as a function of the average pellet diameter d_{pe}, or set a **User defined** value or expression. Either use the **Automatic** option for the **Interstitial heat transfer coefficient**, $h_{\text{pe,f}}$, defined as a function of the average pellets diameter d_{pe} and the fluid-to-solid Nusselt number for which the fluid dynamic viscosity μ is needed (and should be set in the Fluid (Porous Medium) subnode). Or set a **User defined** value or expression for $h_{\text{pe,f}}$.

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options**. The settings of this section have no effect when the **Coupling condition** is **Convective heat flux** in the Pellet-Fluid Interface section.

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.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Porous Media>Porous Medium>Pellet-Fluid Interface

Ribbon
Physics tab with **Porous Medium** selected in the model tree:

Attributes>Pellet-Fluid Interface

Phase Change Material

This subnode should be used to specify the properties of a phase change material according to the apparent heat capacity formulation, when modeling heat transfer in solids, fluids, and porous media. 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. Alternatively, a Stefan condition can be defined on boundaries by using the Phase Change Interface node.

NUMBER OF PHASE TRANSITIONS
To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options** in the **Show More Options** dialog box. 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 an alternative value 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.

DENSITY

This section is only available when the subnode is added under the **Solid** node. In this particular case, a single density should be defined for all phases to ensure mass conservation on the material frame. Default is taken **From material**. For **User defined**, enter a value or expression for ρ. Note that this setting overrides the setting in the **Thermodynamics, Solid** section of the parent **Solid** node.

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.

Open the **Sketch** section for more details on these parameters.

The **Latent heat from phase 1 and phase 2** $L_{1 \to 2}$ should be set to define the latent heat per unit mass released during the first phase 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

$$T_{\text{pc},\,1\to2} < T_{\text{pc},\,2\to3} < \cdots$$

This defines distinct domains of temperature bounded by $T_{\text{pc},\,j-1\to j}$ and $T_{\text{pc},\,j\to j+1}$ 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 $T_{\text{pc},\,j\to j+1} - \Delta T_{j\to j+1}/2$ and $T_{\text{pc},\,j\to j+1} + \Delta T_{j\to j+1}/2$ 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**.

When the **Phase Change Material** subnode is added under a **Solid** node, the following material properties should be set:

- **Thermal conductivity** k_i. The default uses the material values for phase i. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics physics interface for example), you can select on which frame the **Thermal conductivity** k_i is specified, with the **Deformation model for thermal conductivity** option, see Heat Conduction, Solid for details.

- **Heat capacity at constant pressure** $C_{p,\,i}$. The default is 4200 J/(kg·K).

Note that these settings override the settings of the **Heat Conduction, Solid** and **Thermodynamics, Solid** sections of the parent **Solid** node.

When the **Phase Change Material** subnode is added under a **Fluid** or **Porous Medium** node, the following material properties should be set:

- **Thermal conductivity** k_i. The default uses the material values for phase i. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. The default is 1 W/(m·K).

- **Density** ρ_i. The default is 1000 kg/m^3.

- **Heat capacity at constant pressure** $C_{p,i}$. The default is 4200 J/(kg·K).
- **Ratio of specific heats** γ_i. The default is **Automatic**, it calculates the ratio of specific heats by using Mayer's relation:

$$\gamma = \frac{\rho C_p \chi_t}{\rho C_p \chi_t - T\alpha_p^2}$$

considering that:

- α_p is the coefficient of thermal expansion (SI unit: 1/K):

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

- χ_t is the isothermal compressibility (SI unit: 1/Pa):

$$\chi_t = -\frac{1}{\rho}\frac{\partial \rho}{\partial p}$$

Note that these settings override the settings of the **Heat Conduction, Fluid** and **Thermodynamics, Fluid** sections of the parent node. In porous media, phase change is considered in the fluid material only.

🔍	Apparent Heat Capacity Method

💡	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.

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

Phase Change: Application Library path **Heat_Transfer_Module/Phase_Change/phase_change**

Continuous Casting — Apparent Heat Capacity Method: Application Library path **Heat_Transfer_Module/Thermal_Processing/continuous_casting_apparent_heat_capacity**

Cooling and Solidification of Metal: Application Library path **Heat_Transfer_Module/Thermal_Processing/cooling_solidification_metal**

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids>Solid>Phase Change Material
Heat Transfer in Fluids>Fluid>Phase Change Material
Heat Transfer in Porous Media>Porous Medium>Phase Change Material
Heat Transfer in Porous Media>Porous Medium>Fluid>Phase Change Material
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Phase Change Material

Ribbon
Physics tab with **Solid**, **Fluid**, or **Porous Medium** selected in the model tree:

Attributes>Phase Change Material

Physics tab with **Porous Medium>Fluid** selected in the model tree:

Attributes>Phase Change Material

Porous Matrix (Porous Medium, Moist Porous Medium)

This node defines the porosity and material properties of the solid matrix used in the heat transfer equations of the Moist Porous Medium and Porous Medium parent

nodes, to model heat transfer in a porous matrix, possibly consisting of several solids, and filled with a mobile fluid, and one or more immobile fluids.

	• The velocity field and material properties of the mobile fluid can be specified in the Fluid (Porous Medium) subnode, by defining it as a general gas or liquid, as an ideal gas, or as moist air.
	• The total volume fraction and material properties of immobile fluids can be specified in the Immobile Fluids (Porous Medium) subnode.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable minput.Tempref, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (▤), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

Other options are **User defined** and all the temperature variables from the physics interfaces included in the model.

	This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button (▤), and

in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

MATRIX PROPERTIES

The default **Porosity** ε_p of the solid matrix is taken **From material**. In a **Porous Material** node, the porosity is defined as follows:

$$\varepsilon_p = 1 - \sum_i \theta_{si} - \sum_i \theta_{imfi}$$

where θ_{si} and θ_{imfi} are the volume fractions of the **Solid** and **Immobile Fluids** subnodes under the **Porous Material** node.

For **User defined**, enter a value or expression.

Choose to define either the **Dry bulk properties** or the **Solid phase properties** for thermal conductivity, density, and heat capacity at constant pressure of the porous matrix. The first option sets the properties of the matrix including empty pores, while the second option defines the properties of the pure solid phase.

HEAT CONDUCTION, POROUS MATRIX

The thermal conductivity k_s describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k_s \nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k_s is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full**, 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 thermal conductivity is taken **From material** and more than one **Solid** subnode is added under the **Porous Material** node, the average property is obtained by applying a volume average model:

$$k_s = \sum_i \theta_{si} k_{si}$$

Alternatively, set the **Dry bulk thermal conductivity** k_b, when **Define** is set to **Dry bulk properties** in the **Matrix Properties** section. The corresponding property for the solid phase is then defined as:

$$k_s = \frac{k_b}{\theta_s}$$

Note that this section is not available when **Porous medium type** is set to **Local thermal equilibrium** and **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list of the **Porous medium** parent node.

THERMODYNAMICS, POROUS MATRIX

This section defines 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 of mass of the solid material.

The **Density** ρ_s and the **Specific heat capacity** $C_{p,\,s}$ should be specified. For **From Material** option, see Material Density in Features Defined in the Material Frame if a temperature-dependent density should be set.

When the density and specific heat capacity are taken **From material** and more than one **Solid** subnode is added under the **Porous Material** node, the effective properties are obtained by applying a volume averaging model:

$$\rho_s = \sum_i \theta_{si}\rho_{si}$$

$$C_{p,\,s} = \frac{\displaystyle\sum_i \theta_{si}\rho_{si}C_{p,\,si}}{\displaystyle\sum_i \theta_{si}\rho_{si}}$$

Alternatively, set the **Dry bulk density** ρ_b and the **Dry bulk heat capacity at constant pressure** $C_{p,b}$ when **Define** is set to **Dry bulk properties** in the **Matrix Properties** section. The corresponding properties for the solid phase are then defined as:

$$\rho_s = \frac{\rho_b}{\theta_s}$$

and

$$C_{p,\,\mathrm{s}} \; = \; \frac{C_{p,\,\mathrm{b}}}{\theta_\mathrm{s}}$$

> When the porosity ε_p is taken **From material**, the solid volume fraction θ_s is defined in the material, under the **Solid** subnode. In case of multiple solids, it is needed to refer to the material, as it is not possible to deduce the values of $\theta_{\mathrm{s}i}$ for the different solids from the porosity. When the porosity is **User defined**, the volume fraction of each solid $\theta_{\mathrm{s}i}$ is obtained by dividing $\theta_\mathrm{s} = 1 - \varepsilon_\mathrm{p}$ by the number of solids.

> When the **Porous Medium type** is set to **Local thermal nonequilibrium** in the Porous Medium parent node, the Initial Values, Heat Source, Thermal Insulation, Symmetry (Heat Transfer Interface), Temperature, Heat Flux, Lumped System Connector, Continuity, Boundary Heat Source, Surface-to-Ambient Radiation (Heat Transfer Interface), and Deposited Beam Power features are available under the **Porous Matrix** subnode. These subnodes allow the definition of domain and boundary conditions specific to the fluid phase temperature T_s.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Porous Media>Porous Medium>Porous Matrix
Heat Transfer in Moist Porous Media>Moist Porous Medium>Porous Matrix

Ribbon
Physics tab with **Porous Medium** or **Moist Porous Medium** selected in the model tree:

Attributes>Porous Matrix

Porous Medium

This node models heat transfer in a porous matrix, possibly consisting of several solids, and filled with a mobile fluid, and one or more immobile fluids. It allows to make the local thermal equilibrium assumption or not, and to handle the case of a packed bed of pellets in a specific way, through the **Porous medium type** option.

Local thermal equilibrium

With this assumption, one single equation is solved for both phases, using effective material properties. See Equation 4-41 for details.

In particular, when no immobile fluid is present in the pore space, the effective volumetric heat capacity at constant pressure is defined as

$$(\rho C_p)_{\text{eff}} = \theta_s \rho_s C_{p, s} + \varepsilon_p \rho_f C_{p, f}$$

If the porous matrix consists of several solids i of volume fraction θ_{si}, heat capacity $C_{p,si}$, and density ρ_{si}, the above equation is modified as follows:

$$(\rho C_p)_{\text{eff}} = \sum_i \theta_{si} \rho_{si} C_{p, si} + \varepsilon_p \rho_f C_{p, f}$$

See Immobile Fluids (Porous Medium) to take into account the presence of immobile fluids in the expression of effective volumetric heat capacity at constant pressure.

See Porous Medium for the definition of the effective thermal conductivity.

Local Thermal Nonequilibrium

When the solid and fluid temperatures are not in equilibrium, two heat equations are solved in the solid and fluid subdomains, and coupled through a transfer term proportional to the temperature difference between both phases. See Equation 4-43 and Equation 4-44 for details about the heat transfer equation in each phase, and Porous Medium for the definition of the transfer term between fluid and solid phases.

Packed Bed

When the porous medium is a packed bed of pellets, and when the thermal conductivity of the pellets is small enough compared to the fluid conductivity to neglect heat transfer between the pellets, the macroscale heat equation for the solid phase of the standard local thermal nonequilibrium model is replaced by a microscale equation for the radial variation of the temperature due to conduction in the pellets.

See Equation 4-45 and Equation 4-46 for details about the heat transfer equation in each phase, and Pellet-Fluid Interface (Porous Medium) for the definition of the coupling condition between the fluid phase and pellets equations.

COORDINATE SYSTEM SELECTION

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate**

system, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The subnodes of **Porous Medium** inherit these coordinate system settings. In particular, the **Velocity field** and **Thermal Conductivity** (in Fluid (Porous Medium) subnode) and the **Dry bulk thermal conductivity** and **Solid phase thermal conductivity** (in Porous Matrix (Porous Medium, Moist Porous Medium) subnode) should be set according to the coordinate system selected in this section.

See Coordinate Systems in the *COMSOL Multiphysics Reference Manual* for more details.

PODOUS MEDIUM

Select between the **Local thermal equilibrium**, the **Local thermal nonequilibrium**, and the **Packed bed** options in the **Porous medium type** list.

Depending upon the selected option, further settings are required underneath: either the effective thermal conductivity required by the single heat transfer equation, or the parameters used to couple the two heat transfer equations in each phase. See the Local thermal equilibrium and Local Thermal Nonequilibrium sections below for details.

It also changes the availability of some subnodes:

- The **Geothermal Heating**, **Immobile Fluids**, and **Optically Thick Participating Medium** subnodes are available under **Porous Medium** only when the **Porous Medium type** is set to **Local thermal equilibrium**.

- The Initial Values, Heat Source, Thermal Insulation, Symmetry (Heat Transfer Interface), Temperature, Heat Flux, Lumped System Connector, Phase Change Interface, Continuity, Inflow, Outflow, Open Boundary, Boundary Heat Source, Surface-to-Ambient Radiation (Heat Transfer Interface), and Deposited Beam Power features are available under the Fluid (Porous Medium) subnode only when the **Porous Medium type** is set to **Local thermal nonequilibrium** or **Packed bed**. These subnodes allow the definition of domain and boundary conditions specific to the fluid phase temperature T_f.

- The Initial Values, Heat Source, Thermal Insulation, Symmetry (Heat Transfer Interface), Temperature, Heat Flux, Lumped System Connector, Continuity, Boundary Heat Source, Surface-to-Ambient Radiation (Heat Transfer Interface), and Deposited Beam Power features are available under the Porous Matrix (Porous Medium, Moist Porous Medium) subnode only when the **Porous Medium type** is set to **Local thermal nonequilibrium**. These subnodes allow the definition of domain and boundary conditions specific to the solid phase temperature T_s.

Local Thermal Equilibrium

When **Local thermal equilibrium** is selected in the **Porous medium type** list, this section defines the **Effective thermal conductivity** taking into account the properties of the solid matrix and the mobile fluid. The following averaging models are available:

- **Plane layers parallel to heat flow** (default), which calculates the effective conductivity of the solid-fluid system as the weighted arithmetic mean (or volume average) of the conductivities of the fluid and the porous matrix:

$$k_{\text{eff}} = \theta_s k_s + \varepsilon_p k_f$$

- **Plane layers perpendicular to heat flow**, which calculates the effective conductivity of the solid-fluid system as the weighted harmonic mean (or reciprocal average) of the conductivities of the fluid and the porous matrix:

$$\frac{1}{k_{\text{eff}}} = \frac{\theta_s}{k_s} + \frac{\varepsilon_p}{k_f}$$

- **Power law**, which calculates the effective conductivity of the solid-fluid system as the weighted geometric mean of the conductivities of the fluid and the porous matrix:

$$k_{\text{eff}} = k_s^{\theta_s} \cdot k_f^{\varepsilon_p}$$

- **Solid spherical inclusions**, which calculates the effective conductivity of the solid-fluid system as:

$$k_{\text{eff}} = k_f \frac{2k_f + k_s - 2(k_f - k_s)\theta_s}{2k_f + k_s + (k_f - k_s)\theta_s}$$

- **Fluid spherical inclusions**, which calculates the effective conductivity of the solid-fluid system as:

$$k_{\text{eff}} = k_s \frac{2k_s + k_f - 2(k_s - k_f)\varepsilon_p}{2k_s + k_f + (k_s - k_f)\varepsilon_p}$$

- **Wrapped screen**, which calculates the effective conductivity of the solid-fluid system as:

$$k_{\text{eff}} = k_f \frac{k_f + k_s - (k_f - k_s)\theta_s}{k_f + k_s + (k_f - k_s)\theta_s}$$

- **Sintered metal fibers**, which calculates the effective conductivity of the solid-fluid system as:

$$k_{\text{eff}} = \varepsilon_p^2 k_f + \theta_s^2 k_s + \frac{4\varepsilon_p \theta_s k_f k_s}{k_f + k_s}$$

If the porous matrix consists of several solids i of volume fraction θ_{si} and thermal conductivity k_{si}, the above equations are modified by replacing k_s by $\sum_i \theta_{si} k_{si} / \sum_i \theta_{si}$, and θ_s by $\sum_i \theta_{si}$.

ⓧ | The **Immobile Fluids** subnode is available only with the **Plane layers parallel to heat flow**, **Plane layers perpendicular to heat flow**, and **Power law** averaging models.

It is also possible to define directly the effective thermal conductivity, k_{eff}. When **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list, a value for the **Effective thermal conductivity** k_{eff} should be specified directly. The default **Effective thermal conductivity** is taken **From material**. When a **Porous Material** node is active, the property of the **Homogenized Properties** section is used. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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.

Local Thermal Nonequilibrium

When **Local thermal nonequilibrium** is selected in the **Porous medium type** list, select an **Interstitial convective heat transfer coefficient**: **Spherical pellets**, **General configuration**, or **User defined** (the default).

Spherical Pellets In this particular configuration, the interstitial convective heat transfer coefficient can be directly expressed as a function of the average pellets diameter d_{pe} and the fluid-to-solid Nusselt number for which the fluid dynamic viscosity μ is needed.

Enter a value for the **Average diameter** d_{pe} (SI unit: m) of the pellets. Default value is $1e^{-3}$ m.

The dynamic viscosity needed to evaluate the Nusselt number is set in the Fluid (Porous Medium) subnode. See Dynamic Viscosity for details.

Note that with this option, the radial variation of the temperature within the pellets is not precisely accounted for. Change to the **Packed bed** option (in the **Porous medium**

type list) to solve for a specific equation for radial conduction in the pellets. See Packed Bed for details.

General Configuration The interstitial convective heat transfer coefficient is expressed as the product of the specific surface area S_b and the interstitial heat transfer coefficient h_{sf}.

Enter a value for the **Specific surface area** S_b (SI unit: 1/m).

Enter a value for the **Interstitial heat transfer coefficient** h_{sf} (SI unit: W/(m²·K)).

User Defined Enter a custom value for q_{sf} (SI unit: W/(m³·K)).

📝
- See Immobile Fluids (Porous Medium) to take into account the presence of immobile fluids in the expression of effective thermal conductivity in the local thermal equilibrium case.
- The velocity field and material properties of the mobile fluid can be specified in the Fluid (Porous Medium) subnode, by defining it as a general gas or liquid, as an ideal gas, or as moist air.
- The porosity ε_p and material properties of the solid matrix can be specified in the Porous Matrix (Porous Medium, Moist Porous Medium) subnode.
- The packed bed and pellets porosities ε_p and ε_{pe}, and the material properties of the pellets can be specified in the Pellets (Porous Medium) subnode.
- The coupling condition between the fluid phase and the pellets can be specified in the Pellet-Fluid Interface (Porous Medium) subnode.
- When the porosity ε_p is taken **From material**, the solid volume fraction θ_s is defined in the material, under the **Solid** subnode. In case of multiple solids, it is needed to refer to the material, as it is not possible to deduce the values of θ_{si} for the different solids from the porosity. When the porosity is **User defined**, the volume fraction of each solid θ_{si} is obtained by dividing $\theta_s = 1-\varepsilon_p$ by the number of solids.

🔍
See Porous Material in the *COMSOL Multiphysics Reference Manual*.

	Theory for Heat Transfer in Porous Media

	With some COMSOL products, the Thermal Dispersion, Viscous Dissipation, and Geothermal Heating subnodes are available 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 https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Porous Medium

Heat Transfer in Solids>Specific Media>Porous Medium

Heat Transfer in Fluids>Specific Media>Porous Medium

Heat Transfer in Porous Media>Specific Media>Porous Medium

Heat Transfer in Building Materials>Specific Media>Porous Medium

Ribbon

Physics tab with **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, or **Heat Transfer in Building Materials** selected:

Domains>Specific Media>Porous Medium

Pressure Work

This subnode adds the following contribution to the right-hand side of the heat transfer equation to model the result of heating under adiabatic compression as well as some thermoacoustic effects:

$$Q_p = \alpha_p T \left(\frac{\partial p_A}{\partial t} + \mathbf{u} \cdot \nabla p_A \right) \tag{6-13}$$

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 calculated using the relative pressure, and is generally small for low Mach number flows.

No settings are required.

💡	When the **Nonisothermal Flow** multiphysics coupling node is added, the effect of pressure work is automatically taken into account. 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 take into account thermoelastic effects in solids. See Thermoelastic Damping.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Fluids>Fluid>Pressure Work
Heat Transfer in Moist Air>Moist Air>Pressure Work
Heat Transfer in Porous Media>Porous Medium>Fluid>Pressure Work

Ribbon
Physics tab with **Fluid** or **Porous Medium>Fluid** selected in the model tree:

Attributes>Pressure Work

Radiative Source

This node models a radiative heat source within the domain.

When the **Radiation discretization method** is **P1 approximation**, it adds a source term Q_0 to the right-hand side of the equation defining the incident radiation G.

When the **Radiation discretization method** is **Discrete ordinates method**, the source term Q_0 is divided by 4π to obtain the directional power density to be added in the radiative heat transfer equation.

When the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, a source term $Q_{0,k}$ can be defined for each spectral band.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Radiative Source** section are defined in the material or spatial frame:

- The default option for the **Radiative Source** node is **Solid**, which specifies that the heat source Q_0 is defined in the material frame. Because the radiation variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_0 in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

🖹 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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

RADIATIVE SOURCE

Select a **Radiative source type** from the list:

- For **Power density** enter a value for the power density Q_0. When the **Radiation discretization method** is **Discrete ordinates method**, Q_0 is divided by 4π to obtain the directional power density $q_{0,j}$ to be added in the radiative heat transfer equation. When the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the power density can also be **User defined for each band**, or set as a **User defined distribution**.

- For **Directional power density** (available when the **Radiation discretization method** is **Discrete ordinates method**) enter directly a value for the directional power density $q_{0,j}$. When the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the directional power density can also be **User defined for each band**, or set as a **User defined distribution**.

- For **Energy rate** enter a value for the energy rate P_0. In this case $Q_0 = P_0/V$, where V is the total volume of the selected domains. When the **Radiation discretization method** is **Discrete ordinates method**, Q_0 is divided by 4π to obtain the directional power density $q_{0,j}$ to be added in the radiative heat transfer equation. When the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the energy rate can also be **User defined for each band**, or set as a **User defined distribution**.

When **User defined for each band** is selected in any of the options above, set a value for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

When **User defined distribution** is selected in any of the options above, set a value or expression. Any expression is then averaged on each spectral band to obtain a piecewise constant function. If the average value on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

LOCATION IN USER INTERFACE

Context Menus
Radiation in Participating Media>Radiative Source
Radiation in Absorbing-Scattering Media>Radiative Source

Ribbon
Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Domains>Radiative Source

Shape Memory Alloy

This node models heat transfer in shape memory alloys, and takes into account the martensite and austenite composition. This feature is designed to be coupled with the **Shape Memory Alloy** feature of the Structural Mechanics interface which calculates the alloy composition from the mechanical and thermal loads. Changes in martensite and austenite composition modify the alloys thermal properties (thermal conductivity and

heat capacity), and release (from austenite to martensite state) or absorb (from martensite to austenite state) energy.

It uses this version of the heat equation to model heat transfer in alloys:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \tag{6-14}$$

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

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the alloy's density.
- C_p (SI unit: $J/(kg \cdot K)$) is the alloy's heat capacity at constant pressure.
- k (SI unit: $W/(m \cdot K)$) is the alloy's 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 more 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.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature
This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

> ⚠ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

SHAPE MEMORY ALLOY

The **Martensite volume fraction**, ξ, and the **Density** of the alloy (defined for both austenite and martensite states) should be set in this section.

In addition, the following options are available for the computation of the **Effective conductivity** by accounting for both austenite and martensite properties:

- **Volume average** (default), which calculates the effective conductivity of the alloy as the weighted arithmetic mean of austenite and martensite conductivities:

$$k = \xi k_{\text{M}} + (1 - \xi)k_{\text{A}}$$

- **Reciprocal average**, which calculates the effective conductivity of the alloy as the weighted harmonic mean of austenite and martensite conductivities:

$$\frac{1}{k} = \frac{\xi}{k_{\text{M}}} + \frac{1 - \xi}{k_{\text{A}}}$$

- **Power law**, which calculates the effective conductivity of the alloy as the weighted geometric mean of austenite and martensite conductivities:

$$k = k_{\text{M}}^{\xi} \cdot k_{\text{A}}^{(1 - \xi)}$$

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics physics interface for example), you can select on which frame the thermal conductivities of the austenite and martensite phases are specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivities are supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the effective conductivity variables. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivities are supposed to be given on the spatial frame. In case of isotropic materials, the effective thermal conductivity variables are directly equal to the weighted average of the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{(x, y, z)} = \mathbf{R} k_{(X, Y, Z)} \mathbf{R}^{\mathrm{T}}$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

AUSTENITE

Select any component material from the list to define the austenite material properties. The default uses the **Domain material**.

The default **Thermal conductivity** k_{A} is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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 default **Heat capacity at constant pressure** $C_{p,\mathrm{A}}$ is taken **From material**. For **User defined** enter a value or expression.

MARTENSITE

Select any component material from the list to define the martensite material properties. The default uses the **Domain material**.

The default **Thermal conductivity** k_{M} is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** 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 default **Heat capacity at constant pressure** $C_{p,\mathrm{M}}$ is taken **From material**. For **User defined** enter a value or expression.

Context Menus

Heat Transfer in Solids and Fluids>Specific Media>Shape Memory Alloy

Heat Transfer in Solids>Specific Media>Shape Memory Alloy

Heat Transfer in Fluids>Specific Media>Shape Memory Alloy

Heat Transfer in Porous Media>Specific Media>Shape Memory Alloy

Heat Transfer in Building Materials>Specific Media>Shape Memory Alloy

Bioheat Transfer>Specific Media>Shape Memory Alloy

Ribbon

Physics tab with **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Domains>Specific Media>Shape Memory Alloy

Solid

This node uses this version of the heat equation to model heat transfer in solids:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \tag{6-15}$$

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

with the following material properties, fields, and sources:

- ρ (SI unit: $\mathrm{kg/m^3}$) is the solid density.
- C_p (SI unit: $\mathrm{J/(kg{\cdot}K)}$) is the solid heat capacity at constant pressure.
- k (SI unit: $\mathrm{W/(m{\cdot}K)}$) 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 more 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.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature
This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (🖳), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

> ⚠ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature
This section is available when material properties are temperature-dependent. 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), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (🖳), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

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 **Full** 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}.

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics physics interface for example), you can select on which frame the thermal conductivity is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables `ht.kxx`, `ht.kyy`, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables `ht.kxx`, `ht.kyy`, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{(x,\,y,\,z)} \;=\; \mathbf{R} k_{(X,\,Y,\,Z)} \mathbf{R}^{\mathrm{T}}$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

> ⚠️ Fourier's law assumes that the thermal conductivity tensor is symmetric. A nonsymmetric tensor can lead to unphysical results.

This section defines 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`.

	• Theory for Heat Transfer in Porous Media
🔍	• 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.

	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
🔍	https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>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

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Domains>*interface*>Solid

Thermal Damage

This node defines a transformation model to take into account damage by overheating or freezing in biological tissues. The transformation indicator is calculated by a model based either on a temperature threshold or on Arrhenius kinetics.

DAMAGED TISSUE

Three methods are available for the analysis; choose the **Transformation model**: **Temperature threshold** (default), **Arrhenius kinetics**, or **User defined**. Depending on the material properties you have access to, you may choose one of the damage models.

The **Arrhenius kinetics** method is only applicable to hyperthermia analysis.

For **Temperature threshold**, define the settings for the Hyperthermia Analysis or Cryogenic Analysis. See Temperature Threshold for more details on the parameters of the model.

Hyperthermia Analysis
Enter values for:

- **Damage temperature** $T_{d, h}$ to define the (high) temperature that the tissue needs to reach to start getting damaged. The default is 323.15 K.

- **Damage time** $t_{d, h}$ to define the time needed for the necrosis to happen while the temperature is above $T_{d, h}$. The default is 50 s.

- **Necrosis temperature** $T_{n, h}$ to define the (high) temperature to be reached for the necrosis to happen instantly. The default is 373.15 K.

- **Enthalpy change** $L_{d, h}$ to define the enthalpy variation associated with damage by hyperthermia. The following heat source is added to the right-hand side of the bioheat equation:

$$Q = -\rho L_{\mathrm{d,h}} \frac{\partial \theta_{\mathrm{d}}}{\partial t} (T > T_{\mathrm{d,h}})$$

A user defined value can be set for the initial damaged tissue indicator and the initial instant necrosis indicator used in the hyperthermia analysis. See the Initial Values section for details.

Cryogenic Analysis
Enter values for:

- **Damage temperature** $T_{\mathrm{d,c}}$ to define the (low) temperature that the tissue needs to reach to start getting damaged. The default is 273.15 K.

- **Damage time** $t_{\mathrm{d,c}}$ to define the time needed for the necrosis to happen while the temperature is below $T_{\mathrm{d,c}}$. The default is 50 s.

- **Necrosis temperature** $T_{\mathrm{n,c}}$ to define the (low) temperature to be reached for the necrosis to happen instantly. The default is 253.15 K.

- **Enthalpy change** $L_{\mathrm{d,c}}$ to define the enthalpy variation associated with damage by freezing. The following heat source is added to the right-hand side of the bioheat equation:

$$Q = \rho L_{\mathrm{d,c}} \frac{\partial \theta_{\mathrm{d}}}{\partial t} (T < T_{\mathrm{d,c}})$$

A user defined value can be set for the initial damaged tissue indicator and the initial instant necrosis indicator used in the cryogenic analysis. See the Initial Values section for details.

Arrhenius Kinetics
For **Arrhenius kinetics**, define the parameters to calculate the degree of tissue injury with the Arrhenius equation (see Arrhenius Kinetics for more details):

- **Frequency factor** A in the Arrhenius equation. Default is taken **From material**. For **User defined** enter a value or an expression.

- **Activation energy** ΔE in the Arrhenius equation. Default is taken **From material**. For **User defined** enter a value or an expression.

- **Polynomial order** n to define a polynomial Arrhenius kinetics equation.

- **Enthalpy change** L to define the enthalpy variation associated with damage. The following heat source is added to the right-hand side of the bioheat equation:

$$Q= -\rho L\frac{\partial\theta_d}{\partial t}$$

A user defined value can be set for the initial damaged tissue indicator of the Arrhenius equation. See the Initial Values section for details.

User Defined
Enter values or expressions for the **Enthalpy change** L and the **Fraction of transformation** θ_d to define the heat source associated with the transformation as:

$$Q= -\rho L\frac{\partial\theta_d}{\partial t}$$

As required, also define how to Specify Different Material Properties for healthy and damaged tissue.

Specify Different Material Properties
When the **Specify different material properties for damaged tissue** check box is selected, choose a **Transformed 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** and **Thermodynamics** sections. The effective tissue properties change from the healthy tissue properties to the damaged tissue properties as the damage evolves.

HEAT CONDUCTION
This section is available when the **Specify different material properties for damaged tissue** check box is selected.

Select a **Thermal conductivity** k_d — **From material** (default) or **User defined**, to be used for damaged tissue. For **User defined** choose **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity and enter another value or expression in the field or matrix.

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics physics interface for example), you can select on which frame the **Thermal conductivity** k_d is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables ht.k_dxx, ht.k_dyy, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables `ht.k_dxx`, `ht.k_dyy`, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{d(x, y, z)} = \mathbf{R} k_{d(X, Y, Z)} \mathbf{R}^{\mathrm{T}}$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

THERMODYNAMICS

This section is available when the **Specify 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** (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.

INITIAL VALUES

This section is available when the **Transformation model** is set to **Temperature threshold** or **Arrhenius kinetics**. Set a value or expression for the **Initial damaged tissue indicator**, α_{init}, to be used as an initial condition for any of the time integral analyses. For the **Temperature threshold** model, a value or expression for the **Initial instant necrosis indicator**, $\alpha_{n,init}$, can be set as well.

	• Theory for Bioheat Transfer
🔍	• Biological Tissue
	• Bioheat

	There are specific predefined materials available in the Bioheat material
💡	database. See Materials Overview and Bioheat Materials Library 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>Thermal Damage

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Bioheat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media** or **Heat Transfer in Building Materials** selected:

Domains>*interface*>Biological Tissue>Thermal Damage

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 defines the **Longitudinal dispersivity** λ_{lo} and **Transverse dispersivity** λ_{tr} used for the computation of the tensor of dispersive thermal conductivity:

$$(k_{\mathrm{disp}})_{ij} = \rho C_p D_{ij}$$

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

$$\lambda_{ijkl} = \lambda_{\mathrm{tr}}\delta_{ij}\delta_{kl} + \frac{\lambda_{\mathrm{lo}} - \lambda_{\mathrm{tr}}}{2}(\delta_{ik}\delta_{jl} + \delta_{il}\delta_{jk})$$

When the **Horizontal and vertical transverse dispersivities** option is selected in the **Define transverse dispersivity from** list, the **Transverse horizontal dispersivity** $\lambda_{\mathrm{tr,\,h}}$ and **Transverse vertical dispersivity** $\lambda_{\mathrm{tr,\,v}}$ 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

$$D_{ij} =$$

$$\frac{1}{|\mathbf{u}|}\begin{bmatrix} \lambda_{\mathrm{lo}}u^2 + \lambda_{\mathrm{tr,\,h}}v^2 + \lambda_{\mathrm{tr,\,v}}w^2 & (\lambda_{\mathrm{lo}} - \lambda_{\mathrm{tr,\,h}})uv & (\lambda_{\mathrm{lo}} - \lambda_{\mathrm{tr,\,v}})uw \\ (\lambda_{\mathrm{lo}} - \lambda_{\mathrm{tr,\,h}})uv & \lambda_{\mathrm{tr,\,h}}u^2 + \lambda_{\mathrm{lo}}v^2 + \lambda_{\mathrm{tr,\,v}}w^2 & (\lambda_{\mathrm{lo}} - \lambda_{\mathrm{tr,\,v}})vw \\ (\lambda_{\mathrm{lo}} - \lambda_{\mathrm{tr,\,v}})uw & (\lambda_{\mathrm{lo}} - \lambda_{\mathrm{tr,\,v}})vw & \lambda_{\mathrm{tr,\,v}}u^2 + \lambda_{\mathrm{tr,\,v}}v^2 + \lambda_{\mathrm{lo}}w^2 \end{bmatrix}$$

💡	The former formulation corresponds to the general formulation when $\lambda_{\mathrm{tr,\,h}} = \lambda_{\mathrm{tr,\,v}}$.

🔍	Porous Medium

🔍	The **Thermal Dispersion** node is only available with some COMSOL products. For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

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

$$Q_{\text{ted}} = -\alpha T : \frac{\partial S}{\partial t}$$

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*

Attributes>Thermoelastic Damping

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 take that into account. See Equation 4-168 and Equation 4-169.

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.

> The **Thickness** feature is disabled when at least one of the multiphysics couplings (Heat Transfer with Surface-to-Surface Radiation, Heat Transfer with Radiation in Participating Media, Heat Transfer with Radiative Beam in Absorbing Media) is active while solving on the same domain. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

> Out-of-Plane Heat Transfer

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Thickness
Heat Transfer in Solids>Thickness
Heat Transfer in Fluids>Thickness
Heat Transfer in Porous Media>Thickness
Heat Transfer in Building Materials>Thickness
Bioheat Transfer>Thickness

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Domains>*interface***>Thickness**

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 6-15, defined in the parent node:

$$-\rho C_p \mathbf{u}_{\text{trans}} \cdot \nabla T$$

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 $\mathbf{n} \cdot \mathbf{u}_{\text{trans}} \neq 0$. The Heat Flux boundary condition does not, for example, work on boundaries where $\mathbf{n} \cdot \mathbf{u}_{\text{trans}} < 0$.

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** $\mathbf{u}_{\text{trans}}$ 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 take into account 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 **Nonisothermal 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 overridden 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>Porous Medium>Viscous Dissipation
Heat Transfer in Moist Air>Moist Air>Viscous Dissipation
Heat Transfer in Porous Media>Porous Medium>Fluid>Viscous Dissipation

Ribbon
Physics tab with **Fluid**, **Moist Air**, **Porous Medium**, or **Porous Medium>Fluid** selected in the model tree:

Attributes>Viscous Dissipation

Boundary Features

The following boundary nodes and subnodes are available for the Heat Transfer and Heat Transfer in Shells interfaces:

- Boundary Heat Source
- Deposited Beam Power
- External Temperature (Thin Layer, Thin Film, Fracture)
- Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
- Harmonic Perturbation
- Heat Flux
- Heat Source (Heat Transfer in Shells Interface)
- Heat Source (Thin Layer, Thin Film, Fracture)
- Inflow
- Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Isothermal Domain Interface
- Line Heat Source on Axis

- Local Thermal Nonequilibrium Boundary
- Lumped System Connector
- Open Boundary
- Outflow
- Periodic Condition (Heat Transfer Interface)
- Phase Change Interface
- Phase Change Interface, Exterior
- Surface-to-Ambient Radiation (Heat Transfer Interface)
- Symmetry (Heat Transfer Interface)
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in 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.

The following boundary nodes and subnodes are available for the Surface-to-Surface Radiation, Radiation in Participating Media, Radiation in Absorbing-Scattering Media, and Radiative Beam in Absorbing Media interfaces:

- Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Continuity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Diffuse Mirror (Surface-to-Surface Radiation Interface)

- Diffuse Surface (Surface-to-Surface Radiation Interface)

- Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Incident Intensity (Radiative Beam in Absorbing Medium Interface)

- Initial Values (Surface-to-Surface Radiation Interface)

- Layer Opacity (Surface-to-Surface Radiation Interface)

- Opaque Surface (Surface-to-Surface Radiation Interface)

- Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Opaque Surface (Radiative Beam in Absorbing Medium Interface)

- Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Prescribed Radiosity (Surface-to-Surface Radiation Interface)

- Radiation Group (Surface-to-Surface Radiation Interface)

- Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Semitransparent Surface (Surface-to-Surface Radiation Interface)

- Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

- Transparent Surface (Radiative Beam in Absorbing Medium Interface)

For a detailed overview of the functionality available in each product, visit
https://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 **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 to apply this condition to. A pair must to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Boundary Heat Source** section are defined in the material or spatial frame:

- The default option for the **Boundary Heat Source** node is **Solid**, which specifies that the heat source Q_b is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_b in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

BOUNDARY HEAT SOURCE

Select an option from the **Heat Source** list:

- For **General source** (default) 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.

SOURCE POSITION

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Select a **Source position** to define a side where the heat source is defined: **Layer** (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.

📋 You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

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

Evaporative Cooling of Water: Application Library path
Heat_Transfer_Module/Phase_Change/evaporative_cooling

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 in Solids and Fluids>Heat Sources>Boundary Heat Source
Heat Transfer in Solids>Pairs>Boundary Heat Source
Heat Transfer in Porous Media>Porous Medium>Fluid>Heat Sources>Boundary Heat Source
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Heat Sources>Boundary Heat Source

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Boundary Heat Source**
Pairs>*interface***>Boundary Heat Source**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Boundary Heat Source

Continuity

This node can be added to pairs, or as a subnode of the **Fluid** and **Porous Matrix** subnodes under **Porous Medium**. It prescribes that the temperature field is continuous across the boundary. When applied to pairs, **Continuity** is only suitable for pairs where the boundaries match. When added as a subnode of the **Fluid** and **Porous Matrix** subnodes under **Porous Medium**, it ensures the continuity of the temperature and flux between the specific phase (fluid or solid) and the medium on the other side of the boundary (fluid, solid, building material, biological tissue,...). If a thin structure model (Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface), Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface), or Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)) is defined on the boundary, the **Continuity** node may be applied as well.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must 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 in Solids and Fluids>Pairs>Continuity
Heat Transfer in Shells>Pairs>Continuity
Surface-to-Surface Radiation>Pairs>Continuity
Radiation in Participating Media>Pairs>Continuity
Heat Transfer in Porous Media>Porous Medium>Fluid>Continuity
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Continuity

Physics tab with *interface* as **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface, **Heat Transfer in Shells**, **Radiation in Participating Media** or **Surface-to-Surface Radiation** selected:

Pairs> *interface* **>Continuity**

Continuity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node can be added to pairs. When the **Radiation discretization method** is **Discrete ordinates method**, it prescribes that the radiative intensity field I is continuous across the pair. When the **Radiation discretization method** is **P1 approximation**, it prescribes that the incident radiation field G is continuous across the pair. **Continuity** is only suitable for pairs where the boundaries match.

PAIR SELECTION

Choose the pair to apply this condition to. A pair must to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

LOCATION IN USER INTERFACE

Context Menus
Radiation in Participating Media>Pairs>Continuity
Radiation in Absorbing-Scattering Media>Continuity

Ribbon
Physics tab with *interface* as **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Pairs> *interface* **>Continuity**

Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

When **Discrete ordinates method** is selected, this node enables intensity conservation across interior boundaries. It is the default boundary condition for all interior

boundaries. For **P1 approximation**, this boundary condition is not available since it is not needed.

<div style="border: 1px solid;">

▥   *Radiative Heat Transfer in a Utility Boiler*: Application Library path
Heat_Transfer_Module/Thermal_Radiation/boiler

</div>

LOCATION IN USER INTERFACE

Context Menus
Radiation in Participating Media>Continuity on Interior Boundary
Radiation in Absorbing-Scattering Media>Continuity on Interior Boundary

Ribbon
Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Boundaries>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.

BEAM ORIENTATION
Enter a value for the **Beam orientation e**. This vector does not need to be normalized. In 2D axisymmetric components, it is aligned with the z-axis.

BEAM PROFILE
Either select an option for the **Beam profile** among the **Built-in beam profiles**, or set the it as **User defined** by entering a value for the **Beam power density**, I_0 (SI unit: W/m^2). The **Beam power density** corresponds to the beam intensity in the beam cross section.

For **Built-in beam profiles**, enter a value for the **Deposited beam power** P_0 and the coordinates of the **Beam origin point O**.

Then, select a **Distribution type**: **Gaussian** (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.

To display this section click the **Show More Options** button (📷) and select **Advanced Physics Options**.

Select a **Source position** to define a side where the heat source is defined: **Layer** (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 in Solids and Fluids>Heat Sources>Deposited Beam Power
Heat Transfer in Porous Media>Porous Medium>Fluid>Heat Sources>Deposited Beam Power
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Heat Sources>Deposited Beam Power

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Deposited Beam Power**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Deposited Beam Power

Diffuse Mirror (Surface-to-Surface Radiation Interface)

This node is a variant of the Diffuse Surface (Surface-to-Surface Radiation Interface) 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 INPUT

This section contains fields and values that are inputs for expressions defining 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. The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

RADIATION DIRECTION

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

AMBIENT

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

Diffuse Surface (Surface-to-Surface Radiation Interface)

Diffuse surfaces reflect radiative intensity uniformly in all directions. This node handles radiation with a view factor calculation. It is assumed that no radiation is transmitted through the surface. 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

$$-\mathbf{n} \cdot \mathbf{q} = \varepsilon(G - e_b(T))$$

on the side of the boundary where the radiation is defined, where ε is the surface emissivity, G is the irradiation, and $e_b(T)$ 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.

If specular reflection should be considered, use the Opaque Surface (Surface-to-Surface Radiation Interface) node instead.

If specular reflection and transmission should be considered, use the Semitransparent Surface (Surface-to-Surface Radiation Interface) node instead.

If no emission should be considered, use the Diffuse Mirror (Surface-to-Surface Radiation Interface) node instead.

MODEL INPUT

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 is used in the expression of 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.

The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (⧉), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

RADIATION DIRECTION

When **Wavelength dependence of radiative properties** is set to **Constant** in the **Radiation Settings** section of the physics interface, select an **Emitted radiation direction** based on the geometric normal (`nx`, `ny`, `nz`): **Opacity controlled** (default), **Negative normal direction**, **Positive normal direction**, **Both sides**, or **None**.

• **Opacity controlled** requires that each boundary is adjacent to exactly one opaque domain. Opacity is controlled by the Opacity (Surface-to-Surface Radiation Interface) domain feature. For external boundaries, the exterior side opacity is transparent by default but may be edited by setting the **Selection** of the **Opacity** feature on **All voids** in the **Opacity** feature. Figure 6-1 illustrates the emitted radiation direction depending on the opacity of the adjacent domain.
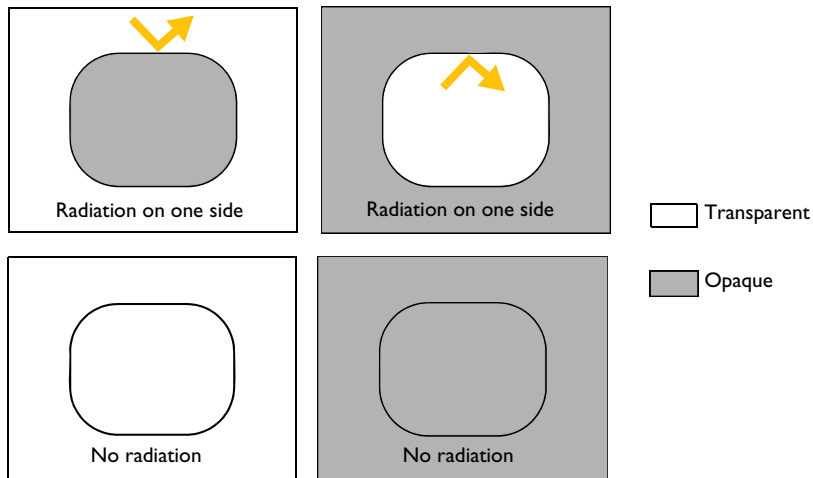
Figure 6-1: Emitted radiation direction with the option Opacity controlled.

- Select **Negative normal direction** to specify that the surface radiates in the negative normal direction (**un** vector direction). An arrow indicates the negative normal direction that corresponds to the direction of the radiation emitted by the surface. Figure 6-2 illustrates the emitted radiation direction depending on the chosen option.

- Select **Positive normal direction** if the surface radiates in the positive normal direction (**dn** vector direction). An arrow indicates the positive normal direction that corresponds to the direction of the radiation emitted by the surface. Figure 6-2 illustrates the emitted radiation direction depending on the chosen option.

- Select **Both sides** if the surface radiates on both sides. Figure 6-2 illustrates the emitted radiation direction depending on the chosen option.

- Select **None** when adjacent domains are either both transparent or both opaque for a given spectral band. When the **Emitted radiation direction** is set to **None** for a spectral band, the radiative properties set for this spectral band are not used
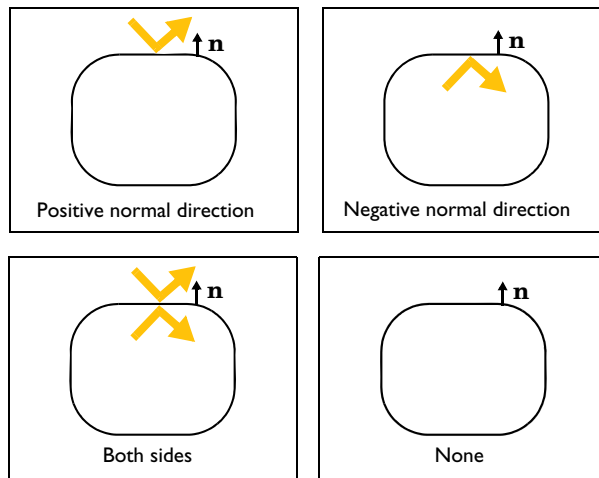
Figure 6-2: Emitted radiation direction with Positive normal direction, Negative normal direction and Both sides options.

When **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands** in the **Radiation Settings** section of the physics interface, select an **Emitted radiation direction** for each spectral band: **Opacity controlled** (default), **Negative normal direction**, **Positive normal direction**, **Both sides**, or **None**. The **Emitted radiation direction** defines the radiation direction for each spectral band similarly as when **Wavelength dependence of radiative properties** 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.

	Note that when **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, the upper bound of the last spectral band, meant to represent the infinite, is set to 1[mm], for the computation of the surface material properties.

AMBIENT

Select the **Define ambient properties on each side** check box when the ambient properties 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 properties on each side** is not selected.

Ambient Temperature
The ambient temperature T_{amb} should be set for the computation of $e_b(T_{amb})$, the ambient blackbody hemispherical total emissive power, used for the evaluation of the ambient irradiation $G_{amb}=F_{amb}\varepsilon_{amb}e_b(T_{amb})$.

Set the **Ambient temperature** T_{amb}. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. When **Define ambient properties on each side** is selected, define the **Ambient temperature, upside** $T_{amb, u}$ and **Ambient temperature, downside** $T_{amb, d}$, respectively. The geometric normal points from the downside to the upside.

	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.

Ambient Emissivity

The ambient emissivity ε_{amb} should be set for the computation of the ambient irradiation $G_{amb}=F_{amb}\varepsilon_{amb}e_b(T_{amb})$.

The **Ambient emissivity** ε_{amb} should be specified. When **Blackbody** is selected, the ambient emissivity is set to 1 for the computation of the ambient irradiation G_{amb}.

If **Wavelength dependence of radiative properties** is **Constant**, choose alternatively the **User defined** option, and set a value or expression. You can define a temperature-dependent emissivity using the variable rad.T.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands,** the ambient irradiation $G_{amb,i}$ of each spectral band i is calculated:

- When **Ambient emissivity** is set to **User defined**, enter a value or expression for the **Ambient emissivity** ε_{amb}. The wavelength may be accessed through the rad.lambda variable. Any expression set for the emissivity is then averaged on each spectral band to obtain a piecewise constant emissivity, with values $\varepsilon_{amb,i}$. If the average value of the emissivity on each band $\varepsilon_{amb,i}$ is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Ambient emissivity** is set to **User defined for each band**, enter a value for the **Ambient emissivity** $\varepsilon_{amb,i}$ for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength.

When **Define ambient properties on each side** is selected, define the **Ambient emissivity, upside** $\varepsilon_{amb, u}$ and **Ambient emissivity, downside** $\varepsilon_{amb, d}$, respectively. The geometric normal points from the downside to the upside.

Diffuse Irradiance

When the **Include diffuse irradiance** check box is selected, a diffuse irradiation contribution I_{diff} is included into the external irradiation. When considering solar irradiation, it takes into account the irradiation from the sun, scattered by the atmosphere, and assumed to be isotropic.

- When **Diffuse irradiance** is set to **User defined**, enter a value or expression for the diffuse irradiance.

- When **Diffuse irradiance** is set to **User defined for each band**, enter a value or expression for the diffuse irradiance for each spectral band.

- Select a **Clear sky noon diffuse horizontal irradiance** defined in an **Ambient Properties** node under **Definitions** to define the diffuse irradiance from ambient properties.

To consider only the direct irradiation defined in the External Radiation Source feature, clear the **Include diffuse irradiance** check box.

FRACTIONAL EMISSIVE POWER

This section is available when the **Wavelength dependence of radiative properties** 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 calculated for each spectral band as a function of the band endpoints and surface temperature.

When the **Fractional emissive power** is **User defined for each band**, define the **Fractional emissive power**, FEP_i 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. Select the **Define fractional emissive power on each side** check box to set specific **Upside** and **Downwside** values in the table.

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.

If **Wavelength dependence of radiative properties** is **Constant**:

- By default, the **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).

- For **User defined**, set a value or expression. You can define a temperature-dependent emissivity using the variable rad.T.

Select the **Define properties on each side** check box to set specific values for the radiative properties on each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside** or **Emissivity, downside** are **From material**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Emissivity** ε (dimensionless) uses values **From material**.

- When **Emissivity** is set to **User defined**, enter a value or expression for the **Emissivity** ε. The wavelength may be accessed through the `rad.lambda` variable. Any expression set for the emissivity is then averaged on each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Emissivity** is set to **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is defined on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each 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.

📝	In the notation used here, `Bi` stands for `B1`, `B2`,… up to the maximum number of spectral intervals.

🔍	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 radiative properties** setting, which is defined for the physics interface. In addition, the **Transparent media refractive index** is equal to 1 by default. 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
Surface-to-Surface Radiation>Diffuse Surface

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries>Diffuse Surface

External Temperature (Thin Layer, Thin Film, Fracture)

Use this subnode to specify the temperature on the exterior side of a thin layer, a thin film, or a fracture, 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 an **Ambient Properties** node under **Definitions**.

The equation for this condition is $T = T_0$ where T_0 is the external temperature to be prescribed.

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

📄	You can specify a harmonic variation of the temperature value in a frequency domain analysis of perturbation type, by adding a **Harmonic Perturbation** subnode. See Theory for Harmonic Heat Transfer for more details.

🔍	• Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) • Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface) • Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids>Thin Layer>External Temperature
Heat Transfer in Solids>Thin Film>External Temperature
Heat Transfer in Solids>Fracture>External Temperature

More locations are available. For example:

Heat Transfer in Fluids>Thin Layer>External Temperature

Ribbon
Physics tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:

Attributes>External Temperature

Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)

Use this node to allow heat transfer in fractures on layered materials inside domains. It can also be used to allow heat transfer in films.

When the **Porous Medium** node is added manually in the Heat Transfer in Fractures interface, and for the **Fracture** node, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

When **Porous Medium** is the default node of the Heat Transfer in Fractures interface, it is applied on all boundaries where the Heat Transfer in Fractures interface is applied, and neither the boundary selection nor the **Restrict to layered boundaries** check box are editable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Restrict to layered boundaries** check box is not editable.

SHELL PROPERTIES

Porous Medium (Heat Transfer in Fractures Interface)
Different settings are available, depending on the settings in the Shell Properties section of the parent interface, and whether **Porous Medium** is the default node or was added manually:

• When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Porous Medium** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers. In this case, both the **General** and **Thermally thin approximation** options are available in the **Layer Model** section.

• When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is taken **From physics interface** in the **Porous Medium** node. This option is not editable when **Porous Medium** is the default node, but you can change to **User defined** and override the interface setting with a specific value or expression for L_{th} when the node was added manually. In this case, only the **Thermally thin approximation** option is available in the **Layer Model** section.

Fracture (Heat Transfer Interface)

Two options are available for the **Shell type**:

- When the **Shell type** is **Layered shell**, the Extra Dimension tool is used to solve the equations through the thickness of a layered material. It is possible to consider several layers with different thermal properties varying through the thickness. This makes the **General** and **Thermally thin approximation** options available for **Layer type** in the **Layer Model** section. You can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- Alternatively, set **Shell type** to **Nonlayered shell**, and set a user defined value or expression for the **Thickness** L_{th}. This option should be used for thermally thin layers, for which no through-thickness temperature variation is expected in the layered material. This removes the **General** option from the **Layer type** list in the **Layer Model** section.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

@
- For a general description of layer and interface selections, see Layer and Interface Selection Tools.
- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

@
The desired selection for the node may correspond to boundaries with different layered materials.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (📑), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

> ⚠️ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (📑), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include 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. The **Common model input** option corresponds to the `minput.pA` variable, set to 1 atm by default. To edit it, click the **Go to Source** button (📑), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section.

LAYER MODEL

The available options for **Layer type** are **Thermally thin approximation** and **General**. The former is a lumped model that accounts only for tangential temperature gradients (along the fracture), whereas the latter accounts also for the normal gradients of temperature (through the fracture's thickness).

> Within a layered material selection, a single **Layer type** should be used. If two layer types are needed for the same layered material, the original material should be duplicated so that one layered material is defined for each layer type. A Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface) node may be added between the two layered materials.

FLUID MATERIAL

By default, the **Boundary material** is used.

HEAT CONDUCTION, FLUID

The settings are the same as for Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface).

HEAT CONDUCTION, FLUID

The settings are the same as for Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface).

THERMODYNAMICS, FLUID

The settings are the same as for Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface).

POROUS MATERIAL

By default, the **Boundary material** is used. The **Volume fraction** θ_p should be specified. The default is 0.

HEAT CONDUCTION, POROUS MATRIX

The default **Thermal conductivity** k_p is taken **From layered material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. Select an **Effective conductivity**: **Plane layers parallel to heat flow** (default) or **Power law**.

The default **Density** ρ_p and **Specific heat capacity** $C_{p,\,p}$ are taken **From layered material**. See Material Density in Features Defined in the Material Frame if a temperature-dependent density should be set. For **User defined** enter other values or expressions.

⚠	In the **Porous Medium** node, only in-plane anisotropy is supported for the thermal conductivity of the fracture material.

🔍	Fracture in Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Porous Media>Thin Structures>Fracture
Heat Transfer in Fractures>Porous Medium

Ribbon
Physics tab with *interface* as **Heat Transfer in Porous Media** selected:

Boundaries>Fracture

Physics tab with *interface* as **Heat Transfer in Shells**, **Heat Transfer in Films** or **Heat Transfer in Fractures** selected:

Boundaries>Porous Medium

Harmonic Perturbation

Use this subnode to specify the harmonic variation of a prescribed temperature value. It is used only in a study step of type **Frequency-Domain Perturbation**.

Enter a value or expression for the **Temperature perturbation** ΔT_0 to be added to the equilibrium temperature T_0 defined in the parent node.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids>Temperature>Harmonic Perturbation

More locations are available. For example:

Heat Transfer in Solids>Thin Rod>Temperature>Harmonic Perturbation
Heat Transfer in Solids>Thin Layer>Temperature>Harmonic Perturbation
Heat Transfer in Solids>Thin Film>Temperature>Harmonic Perturbation
Heat Transfer in Solids>Fracture>Temperature>Harmonic Perturbation
Heat Transfer in Solids>Thin Layer>External Temperature>Harmonic Perturbation
Heat Transfer in Shells>Temperature>Harmonic Perturbation

Ribbon
Physics tab with **Temperature** or **External Temperature** selected in the model tree:

Attributes>Harmonic Perturbation

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 condition instead.

MATERIAL TYPE
Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat flux q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The default option for the **Heat Flux** node is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

📄   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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

HEAT FLUX

Select the **Flux type** from the list: **General inward heat flux** (default), **Convective heat flux**, **Heat rate**, or **Nucleate boiling heat flux**.

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** (default), **Transformer oil**, **Water**, **Moist air**, or **From material**.

When **From material** is selected, choose a material available on the boundary from the **Materials** list.

Depending of the selected option, different parameters are needed. You can refer to the **Sketch** section to get an illustration of the configuration.

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.
- **Characteristic length (area/perimeter)** L. The characteristic length is 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**, **Plate, local transfer coefficient**, **Cylinder in cross flow**, or **Sphere** 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.
- **Cylinder Diameter** D and **Velocity, fluid** U.
- **Sphere Diameter** D and **Velocity, fluid** U.

Internal Forced Convection

For **Internal forced convection** 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 Consistent Stabilization section of a Heat Transfer or Heat Transfer in Shells interface.

External conditions

First, set the **Absolute pressure**, p_A. For **User Defined**, enter a value or expression. Else, select an **Ambient absolute pressure** defined in an **Ambient Properties** node under **Definitions**. The pressure is used to evaluate the **Fluid** material properties and this setting is not available for the **Transformer oil** and **Water** options.

In addition, enter an **External temperature**, T_{ext}. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

Finally, when the **Fluid** is **Moist air**, also set the **External relative humidity**, $\phi_{w, ext}$, and the **Surface relative humidity**, $\phi_{w, s}$, used to evaluate the material properties.

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.

Nucleate Boiling Heat Flux
This option computes q_0 with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See Nucleate Pool Boiling Correlation for details about the correlation.

The correlation applies to clean surfaces, and is insensitive to the shape and orientation of the surface. It relies on empirical constants C_{sf} and s that are predefined for several combinations of fluid and surface materials, and on the boiling fluid properties at saturation temperature, also predefined for some fluids.

Select an option in the **Fluid** list:

- When **Water**, **Benzene**, **n-Pentane**, or **Ethanol** is selected, the material properties of the liquid and vapor phases required in the correlation are predefined (at saturation temperature and p_A=1 atm). Depending on the selected fluid, different materials and finishings are available in the **Surface** list. For each option, the empirical constants C_{sf} and s are predefined.

- When **From material** is selected in the **Fluid** list, first choose materials available on the boundary from the **Liquid materials** and **Vapor materials** lists, to define ρ_l, $C_{p,l}$, k_l, μ_l, and ρ_v. Then set the **Ambient pressure**, p_A, the **Saturation temperature**, T_{sat}, the **Latent heat of evaporation**, L_v, and the **Liquid-vapor surface tension**, σ. Finally, enter values or expressions for the Rohsenow's correlation parameters: the **Prandtl number exponent**, s, and the **Liquid-surface combination factor**, C_{sf}, which accounts for

surface roughness, that tends to increase the number of active nucleation sites for boiling.

Ⓠ

- The detailed definition of the predefined heat transfer coefficients for convective heat transfer is given in Convective Heat Transfer Correlations.
- For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 21.

Ⓠ

- 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 in Solids and Fluids>Heat Flux
Heat Transfer in Porous Media>Porous Medium>Fluid>Heat Flux
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Heat Flux

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Heat Flux**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Heat Flux

Heat Source (Heat Transfer in Shells Interface)

This node describes heat generation within the shell through a $d_\mathrm{s}Q$ contribution to the right-hand side of Equation 4-72. 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.

BOUNDARY SELECTION

Select the boundaries on which to apply the heat source. The **Restrict to layered boundaries** check box makes the node applicable only if a layered material is defined on the boundary. It is editable when the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Heat Source** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is taken **From physics interface** in the **Heat Source** node. You can change to **User defined** and override the interface setting with a specific value or expression for L_th.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

$\mathbb{Q}$	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

	The desired selection for the node may correspond to boundaries with different layered materials.
🔍	See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MATERIAL TYPE

These settings are the same as for the Heat Source node available for the other interfaces.

HEAT SOURCE

These settings are the same as for the Heat Source node available for the other interfaces.

📑	In 2D components the heat source is multiplied by d_z to take into account the out-of-plane thickness.

📑	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

🔍	• Theory for Heat Transfer in Thin Structures • Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells>Heat Source
Heat Transfer in Films>Heat Source
Heat Transfer in Fractures>Heat Source

Ribbon
Physics tab with **Heat Transfer in Shells** selected:

Heat Source (Thin Layer, Thin Film, Fracture)

Use this subnode to add an internal heat source Q_s within a thin layer, thin film, or fracture. Add one or more heat sources.

BOUNDARY SELECTION

By default, the **Heat Source** node is applied on all boundaries where the parent node is applied. See Boundary Selection for details on how to change this setting.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent node:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent node, the **Shell type** is **Layered shell** as well in the **Heat Source** node. This option is not editable, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**, you get access to a list of check boxes for the selection of the individual layers.

- When the **Shell type** to **Nonlayered shell** in the **Shells Properties** section of the parent node, the **Shell type** is taken **From parent feature** in the **Heat Source** node. You can change to **Nonlayered shell** and override the parent node setting with a specific value or expression for the **Thickness** L_{th}.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

🔍	• For a general description of layer and interface selections, see Layer and Interface Selection Tools.
	• You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

The desired selection for the node may correspond to boundaries with different layered materials.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Source** section are defined in the material or spatial frame:

- The default option for the **Heat Source** node is **Solid** when added under **Thin Layer** and **Fracture**, which specifies that the heat source Q_s is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_s in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

HEAT SOURCE

Select the **General source** (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 equal to the volume of the layer selection.

	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

	- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)
	- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
	- Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
	- Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids>Thin Layer>Heat Source
Heat Transfer in Fluids>Thin Film>Heat Source
Heat Transfer in Porous Media>Fracture>Heat Source

More locations are available. For example:

Heat Transfer in Fluids>Thin Layer>Heat Source

Ribbon
Physics tab with **Thin Layer** selected in the model tree:

Attributes>Heat Source

Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

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. It may be specified for each spectral band.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

INCIDENT INTENSITY

When **Wavelength dependence of radiative properties** is **Constant**, the **External radiation intensity** I_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.

If **External radiation intensity** is **User defined**, set a value or expression for I_{ext}.

Alternatively, choose **Blackbody radiation** or **Graybody radiation** to define I_i as $I_b(T_{ext})$ or $\varepsilon I_b(T_{ext})$, respectively (available with the Radiation in Participating Media interface only). Set the **External temperature** T_{ext} (SI unit: K, default value 293.15K) to specify the temperature at which the blackbody radiation intensity I_b is evaluated. Alternatively, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. For **Graybody radiation**, the **Emissivity** ε and **External Temperature** T_{ext} should be defined. If the **Emissivity** is defined **From material**, 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 **Emissivity**, enter another value for ε. If the **Define properties on each side** check box is selected, define the **Emissivity, upside** ε_u and the **Emissivity, downside** ε_d. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside** or **Emissivity, downside** are **From material**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the **External radiation intensity** $I_{i,k}$ for each spectral band should be specified. Select an option between **Graybody radiation**, **Blackbody radiation**, **User defined for each band**, or **User defined distribution**.

If **External radiation intensity** is **Blackbody radiation** or **Graybody radiation** (available with the Radiation in Participating Media interface only), $I_{i,k}$ is defined from the blackbody

intensity $I_b(T_{ext})$, the fractional emissive power for each spectral band at external temperature $FEP_k(T_{ext})$, the emissivity ε_k and the **External temperature** T_{ext} (SI unit: K, default value 293.15K). For **Graybody radiation**, the **Emissivity** ε_k and **External Temperature** T_{ext} should be defined from a user expression or by selecting an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. For **User defined for each band**, set a value for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength. If the **Define properties on each side** check box is selected, define the **Emissivity, upside** ε_u and the **Emissivity, downside** ε_d. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside** or **Emissivity, downside** are **From material**.

If **External radiation intensity** is **User defined for each band**, set a value or expression for $I_{ext,k}$ for each spectral band in the table displayed underneath. If the **Define incident intensity on each side** check box is selected, set values in the **Upside** and **Downside** columns.

If **External radiation intensity** is **User defined distribution**, set a value or expression for $I_{\lambda,ext}$. Any expression is then averaged on each spectral band to obtain a piecewise constant radiation intensity. If the average value of the radiation intensity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average. If the **Define incident intensity on each side** check box is selected, set values for $I_{\lambda,ext,u}$ and $I_{\lambda,ext,d}$.

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 Radiation in Participating Media interface is `rpm` so `rpm.sx`, `rpm.sy`, and `rpm.sz` correspond to the components of discrete ordinate vectors.

- Participating Medium (Radiation in Participating Media Interface)
- Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface)
- Theory for Radiation in Participating Media

Context Menus
Radiation in Participating Media>Incident Intensity
Radiation in Absorbing-Scattering Media>Incident Intensity
Radiation in Participating Media>Pairs>Incident Intensity
Radiation in Absorbing-Scattering Media>Pairs>Incident Intensity

Ribbon
Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Boundaries>Incident Intensity
Pairs>Incident Intensity

Incident Intensity (Radiative Beam in Absorbing Medium Interface)

Use this node to specify an incident radiative beam on the boundaries of an absorbing medium. The incident beam is defined from its propagation direction and incident intensity. See Absorbing Medium (Radiative Beam in Absorbing Medium Interface) for the modeling of the beam absorption in the medium.

BOUNDARY SELECTION
Select the boundaries on which to apply the incident beam. The radiative intensity corresponding to the incident beam will be defined on any selected boundary such that the dot product between the beam orientation and the outgoing normal vector from the physics interface selection is negative.

MODEL INPUT
This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

BEAM ORIENTATION
Enter values for the **Beam orientation e**. The orientation vector does not need to be normalized.

Either select an option for the **Beam profile** among the **Built-in beam profiles**, or set the
it as **User defined** by entering a value for the **Beam power density**, I_0 (SI unit: W/m^2).
The **Beam power density** corresponds to the beam intensity in the beam cross section.

For **Built-in beam profiles**, enter a value for the **Deposited beam power** P_0 and the
coordinates of the **Beam origin point O**.

Then, select a **Distribution type**: **Gaussian** (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.

🗎	This feature does not take into account shielding effect between multiple absorbing media. In addition, a radiative beam leaving an absorbing medium will not generate an incident beam in an absorbing medium placed behind the first one.

🗎	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 heat source location. Otherwise, an automatic substitution replaces the standard deviation σ or the 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*.

🔍	Radiative Beam in Absorbing Media Theory

Context Menus
Radiative Beam in Absorbing Media>Incident Intensity

Ribbon
Physics tab with **Radiative Beam in Absorbing Media** selected:

Boundaries>Incident Intensity

Inflow

Use this node to model inflow of heat through a virtual domain, at inlet boundaries of non-solid domains. The upstream temperature T_{ustr} and upstream absolute pressure p_{ustr} are known at the outer boundary of the virtual domain. The inflow condition is often similar to a temperature condition. However using this condition the upstream temperature is not strictly enforced at the inlet. The higher the flow rate, the smallest difference between the inlet temperature and upstream temperature. In case of heat sources or temperature constraints close to the inlet boundary, the inflow condition induces a temperature profile that approaches to the temperature profile that would be obtained by computing the solution in the virtual domain upstream the inlet.

This boundary condition estimates the heat flux through the inlet boundary from a Danckwerts condition on the enthalpy:

$$-\mathbf{n} \cdot \mathbf{q} = \rho \Delta H \mathbf{u} \cdot \mathbf{n} \qquad (6\text{-}16)$$

where ΔH is the sensible enthalpy. See Theory for the Inflow Boundary Condition for the definition of ΔH and more details on the Danckwerts condition.

UPSTREAM PROPERTIES
The **Upstream temperature** T_{ustr} should be specified. For **User Defined**, enter a value or expression. Else, select any available input (like **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**).

Select the **Specify upstream absolute pressure** check-box to set a different value than the absolute inlet pressure for the **Upstream absolute pressure** p_{ustr}, and take into account work done by pressure forces in the expression of the enthalpy. For **User Defined**, enter a value or expression. Else, select an **Ambient absolute pressure** defined in an **Ambient Properties** node under **Definitions**.

Context Menus
Heat Transfer in Solids and Fluids>Flow Conditions>Inflow
Heat Transfer in Porous Media>Porous Medium>Fluid>Flow Conditions>Inflow
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Flow Conditions>Inflow

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Inflow**

Physics tab with **Porous Medium>Fluid** selected in the model tree:

Attributes>Inflow

Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in 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.

BOUNDARY SELECTION

When the **Initial Values** node is added manually in the Heat Transfer in Shells interface or under the **Thin Layer**, **Thin Film**, or **Fracture** node, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

When **Initial Values** is the default node of the Heat Transfer in Shells interface, or the default subnode of the **Thin Layer**, **Thin Film**, or **Fracture** node, it is applied on all the boundaries where the parent node is applied, and neither the boundary selection nor the **Restrict to layered boundaries** check box are editable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Restrict to layered boundaries** check box is not editable.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent interface or node, and whether **Initial Values** is the default node or was added manually:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface or node, the same layered material is used in the **Initial Values** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box, when this is not the default **Initial Values** node. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Thickness** is taken **From physics interface** or **From parent feature** in the **Initial Values** node. This option is not editable when **Initial Values** is the default node or subnode, but you can change to **User defined** and override the parent setting with a specific value or expression for L_{th} when the node was added manually.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

| | - For a general description of layer and interface selections, see Layer and Interface Selection Tools.
| | - You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates. |

| | The desired selection for the node may correspond to boundaries with different layered materials.
| | See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*. |

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 an **Ambient Properties** node under **Definitions**.

When **Initial Values** is added as a subnode of **Thin Layer**, **Thin Film**, or **Fracture** node with **Layer type** set as **Thermally thick approximation**, it is applicable only on exterior boundaries and defines the initial temperature on the outer side of the thin structure. Note that the initial temperature of the inner side of the thin structure is always defined from the initial condition of the adjacent domains.

When **Initial Values** is added as a subnode of **Thin Layer**, **Thin Film**, or **Fracture** node with **Layer type** set as **Thermally thick approximation** or **General**, an additional option, **From adjacent domains**, is available. When it is selected, the initial temperature of the layer is uniform through the thickness and is equal to the average value of initial temperature in the adjacent domains. Note that this option is not available when the **Initial Values** feature is added in the 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 Solids>Thin Layer>Initial Values
Heat Transfer in Solids>Thin Film>Initial Values
Heat Transfer in Solids>Fracture>Initial Values
Heat Transfer in Shells>Initial Values
Heat Transfer in Films>Initial Values
Heat Transfer in Fractures>Initial Values

Ribbon
Physics tab with **Thin Layer** selected in the model tree:

Attributes>Initial Values

Physics tab with **Heat Transfer in Shells** selected:

Boundaries>Heat Transfer in Shells>Initial Values

Initial Values (Surface-to-Surface Radiation Interface)

This node adds an initial value for the surface radiosity.

INITIAL VALUES

Depending on the **Wavelength dependence of radiative properties** set in the **Radiation Settings** section of the interface, further settings display underneath.

If the **Wavelength dependence of radiative properties** is **Constant**, set the **Initial value** to **Blackbody/Graybody** or **User defined**:

- When **Initial value** is set to **User defined**, enter a value or expression for the **Initial surface radiosity** J. Select the **Define initial surface radiosity on each side** check box to set specific values for the **Initial surface radiosity, upside** J_u and **Initial surface radiosity, downside** J_d.

- When **Radiative intensity** is set to **Blackbody/Graybody**, enter a value for the **Initial temperature** T_{init}.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, set the **Initial value** to **Blackbody/Graybody**, **User defined for each band**, or **User defined**:

- When **Initial value** is set to **User defined**, enter a value or expression for the **Initial surface radiosity distribution** J_λ. The wavelength may be accessed through the `rad.lambda` variable. This distribution is integrated on each spectral band to obtain the source heat rate J_i for each spectral band.

- When **Initial value** is set to **Blackbody/Graybody**, enter a value for the **Initial temperature** T_{init}. For **User defined**, enter a value or expression for the initial value of the **Initial temperature** T_{init}. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

- When **Initial value** is set to **User defined for each band**, enter a value for the **Initial surface radiosity** J_i for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same radiosity is defined on both sides. Select the **Define initial surface radiosity on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

   🔍   Theory for Surface-to-Surface Radiation

Context Menus
Surface-to-Surface Radiation>Initial Values

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries>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.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature
This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must 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** (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 6-5: 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** (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.

When the **Heat transfer coefficient** h option is selected, 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**. A complete description of these options can be found in Heat Flux, Convective Heat Flux.

The **External temperature** found under the section of the same name must be set when the isothermal domain interface is also an exterior boundary.

Nucleate Boiling Heat Flux

The heat flux q_0 is computed with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See Nucleate Pool Boiling Correlation for details about the correlation. Select materials from the **Fluid** and **Surface** lists. When a predefined material (**Water**, **Benzene**, **n-Pentane**, or **Ethanol**) is used for the fluid, no additional parameters are required, because the liquid and vapor properties are predefined (at saturation temperature) and the Rohsenow's correlation parameters C_{sf} and s are available for different surface materials. Else, see Nucleate Boiling Heat Flux (Heat Flux feature) for details about the settings.

Thermal Contact

This condition specifies the conductive heat flux at the interface of a solid isothermal domain adjacent to a solid.

Either specify the **Surface thermal resistance**, $R_{t,s}$ (SI unit: K·m^2/W), or set the **Thermal resistance**, R_t (SI unit: K/W) and evaluate $R_{t,s}$ from it:

$$R_{t,\,s} = R_t A$$

where A (SI unit: m^2) is the surface area of the boundary.

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 calculate 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 an **Ambient Properties** node under **Definitions**.

INTERFACE OPACITY

Select **Opaque** (default) or **Transparent** to set the interface's opacity type.

This is needed when the interface contributes with any boundary condition from the **Radiation** menu. It picks the side where irradiation starts from.

🔍	• Isothermal Domain • Theory for Lumped Isothermal Domain • The detailed definition of the predefined heat transfer coefficients is given in Convective Heat Transfer Correlations.

🔍	• For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 21.

▥	*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 in Solids and Fluids>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 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

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Boundaries>*interface***>Isothermal Domain Interface**

Pairs>*interface***>Isothermal Domain Interface**

Layer Opacity (Surface-to-Surface Radiation Interface)

This subnode enables to define the opacity of a boundary. When the boundary represents a thin layer, this subnode can be used to specify the in-depth opacity of the layer, independently on the opacity at the surface of the layer when using the Diffuse Mirror (Surface-to-Surface Radiation Interface), Diffuse Surface (Surface-to-Surface Radiation Interface), and Opaque Surface (Surface-to-Surface Radiation Interface) nodes.

When several spectral bands are defined, the opacity can be set for all or only a selection of wavelengths.

OPACITY

Depending on the **Wavelength dependence of radiative properties** defined in the Radiation Settings section of the physics interface settings, different sections display underneath:

• When **Wavelength dependence of radiative properties** is set to **Constant**, select **Opaque** or **Transparent** to set the layer's opacity type.

• When **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, set the opacity for each spectral band by selecting the corresponding check box in the **Opaque** column of the table. By default the check box is selected for all spectral bands.

Context Menus

Surface-to-Surface Radiation>Diffuse Mirror>Layer Opacity
Surface-to-Surface Radiation>Diffuse Surface>Layer Opacity
Surface-to-Surface Radiation>Prescribed Radiosity>Layer Opacity

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected in the model tree:

Surface-to-Surface Radiation>Diffuse Surface>Layer Opacity

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.

> You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>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

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Boundaries>*interface***>Line Heat Source on Axis**

Local Thermal Nonequilibrium Boundary

This node is active by default on all the interior and exterior boundaries of the computational domain, adjacent to domains where a **Porous Medium** node is applied with **Porous medium type** set to **Local thermal nonequilibrium**. It shows which boundaries are adjacent to domains where two temperatures, T_f and T_s, are solved.

On these boundaries, the conditions available at the physics interface level are not applicable, because they define conditions on a single temperature, T. There are however a few exceptions on interior boundaries:

- The thin structures conditions (**Thin Layer**, **Thin Film**, **Fracture**, and **Thermal Contact**), which represent a thin domain as a boundary, where a single temperature, T, is solved. They can be combined with other boundary conditions on T, available at the physics interface level on boundaries where a thin structure condition is applied: **Boundary Heat Source**, **Deposited Beam Power**, and **Surface-to-Ambient Radiation**.

- The **Thermal Insulation** node, which enforces thermal insulation for both the fluid and solid phases.

- The **Periodic Condition** node, which applies a periodic condition for each phase.

Other boundary conditions can be applied specifically to the fluid or porous matrix phase, by adding subnodes to the Fluid (Porous Medium) and Porous Matrix (Porous Medium, Moist Porous Medium) subnodes.

Concerning the domain conditions, the **Heat Source** and **Initial Values** nodes, available at the physics interface level, are not applicable in the domains delimited by the **Local Thermal Nonequilibrium Boundary** condition.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Local Thermal Nonequilibrium Boundary
Heat Transfer in Solids>Local Thermal Nonequilibrium Boundary
Heat Transfer in Fluids>Local Thermal Nonequilibrium Boundary
Heat Transfer in Porous Media>Local Thermal Nonequilibrium Boundary
Bioheat Transfer>Local Thermal Nonequilibrium Boundary

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Boundaries>*interface***>Local Thermal Nonequilibrium Boundary**

Lumped System Connector

Use this node to connect the distributed finite element model to a thermal network.

The average temperature on the selected boundaries can be prescribed at a node of the thermal network where an External Terminal feature of The Lumped Thermal System Interface is applied.

In return, the heat rate defined at the node of the thermal network is applied on the selected boundaries.

TERMINAL INPUTS

Select an **External Terminal** feature in the **Source** list to define which node of the thermal network provides the heat rate P_{ext} to be prescribed on the boundary.

When the **Isothermal terminal** check box is not selected (default), the average temperature on the boundaries is constrained to the temperature at the terminal node of the thermal network. Select this check box to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.

▥	*Lumped Composite Thermal Barrier*: Application Library path **Heat_Transfer_Module/Tutorials,_Thin_Structure/ lumped_composite_thermal_barrier**

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Lumped System Connector
Heat Transfer in Porous Media>Porous Medium>Fluid>Lumped System Connector
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Lumped System Connector

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Lumped System Connector**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Lumped System Connector

Opaque Surface (Surface-to-Surface Radiation Interface)

This node is a variant of the Diffuse Surface (Surface-to-Surface Radiation Interface) node with specular reflectivity. It is available when the **Surface-to-surface radiation method** is set to **Ray shooting** in the settings of the Surface-to-Surface Radiation interface. It applies to glossy surfaces where specular radiation should be considered in addition to diffuse radiation. The node adds to its selection a radiosity shape function for each spectral band, and uses it as surface radiosity.

If only diffuse reflection is to be taken into account, use the Diffuse Surface (Surface-to-Surface Radiation Interface) node instead.

It is assumed that no radiation is transmitted through the surface. If radiative transmission is also to be taken into account, use the Semitransparent Surface (Surface-to-Surface Radiation Interface) node instead.

MODEL INPUT

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 is used in the expression of 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.

The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

COORDINATE SYSTEM SELECTION

Select a boundary system from the **Coordinate system** list to define the reference vector for the azimuthal angle. The first component of the boundary system selected is the reference vector used for directional surface properties. The default is the boundary

system of the model, and the list contains any additional boundary system added under the **Definitions** node.

See Coordinate Systems in the *COMSOL Multiphysics Reference Manual* for more details.

RADIATION DIRECTION

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

AMBIENT

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

FRACTIONAL EMISSIVE POWER

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

SURFACE RADIATIVE PROPERTIES

Directional Dependence
By default, the **Directional dependence** of the surface emissivity is set to **Constant**. The **Emissivity** can be defined by an expression depending on other variables such as temperature or spatial coordinates. Switch to the **Polar** or **Polar and azimuthal** options to define the dependences on the polar and azimuthal angles, θ and φ, by specifying a **Directional emissivity function**, f_ε. When using the **Polar** option, select from the **Directional emissivity function** list any function of one argument already defined in the component, under the **Definitions** node. For the **Polar and azimuthal** option, select from the functions of two arguments. By default, only the **Zero** function is available in the list, defining $f_\varepsilon(\theta) = 0$ or $f_\varepsilon(\theta, \varphi) = 0$.

The other dependencies (temperature T, spatial coordinates $\mathbf{x}$) are treated separately in the **Emissivity** expression, and used to define the total emissivity:

$$\varepsilon_{tot}(\theta, \varphi, T, \mathbf{x}) = f_\varepsilon(\theta, \varphi) + \varepsilon(T, \mathbf{x})$$

Select the **Define properties on each side** check box to define specific directional dependences of the surface emissivity on each side of the boundary, and set the **Directional dependence, upside** and **Directional dependence, downside** accordingly.

> 📑 The reference vector for the azimuthal angle is given by the first component of the **Coordinate system** selected in the **Coordinate System Selection** section.

Emissivity, Directional Emissivity Function, and Diffuse Reflectivity
If the **Wavelength dependence of radiative properties** is **Constant**:

- By default, **Emissivity** ε (dimensionless) and **Diffuse reflectivity** ρ_d (dimensionless) use values **From material**. These are properties of the surface that depend 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).

- For **User defined**, set values or expressions. You can define the emissivity and reflectivity as functions of temperature using the variable `rad.T`.

Select the **Define properties on each side** check box to define specific values for each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside**, **Emissivity, downside**, **Diffuse reflectivity, upside**, and **Diffuse reflectivity, downside** are **From material**.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Emissivity** ε (dimensionless) and **Diffuse reflectivity** ρ_d (dimensionless) use values **From material**.

- When **Emissivity** is **User defined**, enter a value or expression for **Emissivity** ε. The wavelength is accessible through the `rad.lambda` variable. Any expression set for the emissivity is then averaged over each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Diffuse reflectivity** is **User defined**, enter a value or expression for the **Diffuse reflectivity** ρ_d. The wavelength is accessible through the `rad.lambda` variable. Any expression set for the reflectivity is then averaged on each spectral band to obtain a piecewise constant reflectivity. If the average value of the reflectivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Emissivity** is **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is set on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

- When **Diffuse reflectivity** is **User defined for each band**, enter a value for the **Diffuse reflectivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same reflectivity is defined on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

Set the surface emissivity to a number between 0 and 1, where 0 represents a diffuse mirror and 1 is suitable for a perfect blackbody. The appropriate value for a physical material is somewhere in between and can be found in tables or measurements.

Set the diffuse reflectivity of the surface to a number between 0 and 1, where 1 applies to a perfect diffuse surface and 0 is appropriate when the reflection is only specular.

The specular surface reflectivity, ρ_s, is then defined as

$$\rho_s = 1 - \rho_d - \varepsilon$$

When the **Emitted radiation direction** is set to **None** for a spectral band, the information defined for that spectral band in this section is not used.

LOCATION IN USER INTERFACE

Context Menus
Surface-to-Surface Radiation>Opaque Surface

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries>Surface-to-Surface Radiation>Opaque Surface

Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node defines a boundary that is opaque to radiation. It prescribes the incident intensities on a boundary and takes into account the net radiative heat flux, $q_{r,net}$, which is absorbed by the surface.

The net radiative heat flux is defined as the difference between the incoming and outgoing radiative heat fluxes:

$$q_{r,\,net} = q_{r,\,in} - q_{r,\,out}$$

The incoming and outgoing radiative heat fluxes are defined from the weighted sums of the incident intensities.

See Opaque Surface for more details on the definition of the radiative intensities on the boundary.

If the boundary is semitransparent, consider using the Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) node instead.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

MODELS INPUT

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 expression of the blackbody radiative intensity.

	The definition of the boundary temperature may be different from that of the temperature in the adjacent domain.

FRACTIONAL EMISSIVE POWER

This section is available when the **Wavelength dependence of radiative properties** is defined as **Solar and ambient** or **Multiple spectral bands** in the Radiation in Participating Media interface (see Participating Media Settings).

When the **Fractional emissive power** is **Blackbody/Graybody**, the fractional emissive power FEP_k is automatically calculated for each spectral band as a function of the band endpoints and temperature.

When the **Fractional emissive power** is **User defined for each band**, define the **Fractional emissive power**, FEP_k for each spectral band in the table displayed underneath. All

fractional emissive powers are expected to be in [0,1] and their sum must be equal to 1. Select the **Define fractional emissive power on each side** check box to set specific **Upside** and **Downwside** values in the table.

SURFACE RADIATIVE PROPERTIES

Select a **Surface type** to define the behavior of the surface: **Gray surface** or **Black surface**. The definition of the incident intensities used to define the incoming and outgoing radiative heat fluxes depends on the option selected in this list.

Gray Surface

If **Gray surface** is selected, the surface emissivity should be set. This is a property of the material surface that depends both on the material itself and the structure of the surface.

An emissivity of 0 means that the surface does not emit any radiation and that all outgoing radiation is diffusely reflected by this boundary. An emissivity of 1 means that the surface is a perfect blackbody: all outgoing radiation is fully absorbed on this boundary. When the node is added in the Radiation in Participating Media interface with the Discrete Ordinates Method, the radiative intensity along the discrete incoming directions on this boundary is defined by

$$I_i = \varepsilon I_b(T) + \frac{1 - \varepsilon}{\pi} q_{r,\,out}$$

When it is added in the Radiation in Absorbing-Scattering Media interface with the Discrete Ordinates Method, no emission is taken into account, and the radiative intensity along the incoming discrete directions on this boundary is defined by

$$I_i = \frac{1 - \varepsilon}{\pi} q_{r,\,out}$$

The default value of the **Emissivity** ε is taken **From material**. Make sure a material is defined at the boundary level (by default materials are defined at the domain level).

For **User defined** enter another value or expression. You can define a temperature-dependent emissivity using the variable `rpm.T` or `rasm.T`.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength can be accessed by the `rpm.lambda` or `rasm.lambda` variable. Any expression defined for the emissivity is then averaged over each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on

each band is known, you may use instead the option **User defined for each band** to avoid the evaluation of the average.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, alternatively set the **Emissivity** to **User defined for each band** to enter a value for each spectral band in the table shown below. Within a spectral band, each value is assumed to be independent of wavelength.

Select the **Define properties on each side** check box to set specific values on each side of the boundary. The **Emissivity, upside** and **Emissivity, downside** should be set, with the same options as those described above.

Black Surface

If **Black surface** is selected, no user input is required, and the radiative intensity along the discrete incoming directions on this boundary is defined by

$$I_i = I_{\mathrm{b}}(T)$$

if the node is added in the Radiation in Participating Media interface. When the node is added in the Radiation in Absorbing-Scattering Media interface,

$$I_i = 0$$

Values of radiative intensity along discrete outgoing 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.

	• Participating Medium (Radiation in Participating Media Interface) • Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface) • Theory for Radiation in Participating Media

	Radiative Heat Transfer in a Utility Boiler: Application Library path **Heat_Transfer_Module/Thermal_Radiation/boiler**

Context Menus
Radiation in Participating Media>Opaque Surface
Radiation in Absorbing-Scattering Media>Opaque Surface
Radiation in Participating Media>Pairs>Opaque Surface
Radiation in Absorbing-Scattering Media>Pairs>Opaque Surface

Ribbon
Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Boundaries>Opaque Surface
Pairs>Opaque Surface

Opaque Surface (Radiative Beam in Absorbing Medium Interface)

Use this node to set an absorbing medium's boundary opaque to radiation. This condition applies to all the radiative beams defined by an active **Incident Intensity** condition. If the beam propagates from the opaque surface to the absorbing medium, the radiative intensity is set to 0. Otherwise, the radiative beam is absorbed by the surface and deposits all its energy on it. A boundary heat source, Q_b, is defined and corresponds to the energy deposited by the entire radiative beam on the opaque surface. See Absorbing Medium (Radiative Beam in Absorbing Medium Interface) for modeling the absorption of the radiative beam in the medium.

Radiative Beam in Absorbing Media Theory

Context Menus
Radiative Beam in Absorbing Media>Opaque Surface

Ribbon
Physics tab with **Radiative Beam in Absorbing Media** selected:

Boundaries>Opaque Surface

Open Boundary

Use this condition to define a boundary as the limit between a non-solid domain of the geometry and the rest of the same fluid domain that is not represented in the geometry. At the open boundary both inflow and outflow conditions are supported.

For an outgoing fluid flow across the boundary, it applies a zero conductive flux condition:

$$-\mathbf{n} \cdot \mathbf{q} = 0, \qquad \mathbf{n} \cdot \mathbf{u} \geq 0 \qquad\qquad (6\text{-}17)$$

For an incoming flow of velocity field $\mathbf{u}$ across the boundary, it accounts by default for the heat flux induced by the flow rate through a Danckwerts condition on the enthalpy:

$$-\mathbf{n} \cdot \mathbf{q} = \rho \Delta H \mathbf{u} \cdot \mathbf{n}, \qquad \mathbf{n} \cdot \mathbf{u} < 0 \qquad\qquad (6\text{-}18)$$

where ΔH is the sensible enthalpy. See Theory for the Inflow Boundary Condition for more details on the Danckwerts condition and the definition of ΔH.

Alternatively the open boundary condition can set a constraint on the temperature for an incoming flow

$$T = T_{\text{ustr}}, \qquad \mathbf{n} \cdot \mathbf{u} < 0 \qquad\qquad (6\text{-}19)$$

Both conditions are based on the knowledge of the upstream temperature at the outer boundary of the virtual domain, but the first one should be preferred to take into account the feedback of the model heat sources and temperature constraints on the inlet temperature profile.

UPSTREAM PROPERTIES

Enter the **Upstream temperature** T_{ustr} outside the open boundary. For **User Defined**, enter a value or expression. Else, select any available input (like an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**).

INFLOW CONDITION

Unfold this section to change the inflow condition:

- The **Flux (Danckwerts)** condition (default) prescribes the flux defined from the upstream temperature and the fluid velocity at the boundary. Using this condition the temperature condition is not strictly enforced at the inlet. Instead a flux is set on

this boundary. The higher the flow rate, the smallest difference between the inlet temperature and upstream temperature.

- The **Discontinuous Galerkin constraints** option enforces the upstream temperature on the 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 in Solids and Fluids>Flow Conditions>Open Boundary
Heat Transfer in Porous Media>Porous Medium>Fluid>Flow Conditions>Open Boundary
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Flow Conditions>Open Boundary

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Open Boundary**

Physics tab with **Porous Medium>Fluid** selected in the model tree:

Attributes>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.

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 in Solids and Fluids>Flow Conditions>Outflow
Heat Transfer in Porous Media>Porous Medium>Fluid>Flow Conditions>Outflow
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Flow Conditions>Outflow

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Outflow**

Physics tab with **Porous Medium>Fluid** selected in the model tree:

Attributes>Outflow

Periodic Condition (Heat Transfer Interface)

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**

Context Menus

Heat Transfer in Solids and Fluids>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

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**,
Heat Transfer in Fluids, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**
or **Bioheat Transfer** selected:

Boundaries>*interface*>Periodic Condition

Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

Use this node to add periodic conditions for the radiative intensity or incident
radiation on 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*.

LOCATION IN USER INTERFACE

Context Menus
Radiation in Participating Media>Periodic Condition
Radiation in Absorbing-Scattering Media>Periodic Condition

Ribbon
Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering
Media** selected:

Boundaries>Periodic Condition

Phase Change Interface

Use this node to define the temperature and the position of the interface between two phases of the same material, using the Stefan condition. It is applicable on boundaries between Fluid and Solid domains, and on interior boundaries of Fluid, Solid, and Porous Medium domains, in 2D, 2D axisymmetric, and 3D components.

For an equivalent condition applicable on exterior boundaries, use Phase Change Interface, Exterior instead.

The following hypotheses are made when using this node:

- One of the phases, the one that corresponds to the coldest phase, should be immobile. The properties of this phase are denoted by subscript s, which stands for solid, but this phase can be an immobile fluid as well. The properties of the other phase are denoted by subscript f, which stands for fluid.

- In a porous medium, the porous matrix is not subject to phase change, which occurs only in the fluid material. The effective density of the phase change material used in the Stefan condition depends on the porosity. Thus, a Porous Medium node should be active on each side of the phase change interface.

- Mass conservation should be verified. This can be ensured by using a velocity field calculated by a Fluid Flow interface in the fluid domain. If Solid nodes are used for both phases, they should have the same density, to ensure mass conservation on the material frame.

- No topology changes are allowed for the interface. If some topology change is expected, the Phase Change Material subnode should be used instead.

In stationary and time-dependent studies, the temperature is set to the phase change temperature on the interface:

$$T = T_{pc}$$

In addition, in time-dependent studies, the Stefan condition defines the phase change interface velocity v_n from the conductive heat flux jump across the interface, $\mathbf{q}$, the latent heat of phase change from solid to fluid, $L_{s \to f}$, and the solid density, ρ_s:

$$v_n = \frac{\mathbf{q}}{\rho_s L_{s \to f}}$$

with

$$\mathbf{q} = -k_s \nabla T_s + k_f \nabla T_f$$

The fluid velocity at the interface, v_f, is defined as:

$$v_f = \frac{\rho_f - \rho_s}{\rho_f} v_n$$

In order to simulate the deformation of the fluid and solid domains, a **Deforming Domain** node (under the **Deformed Geometry** branch in **Definitions**) should be active on the adjacent domains on each side of the phase change interface, the velocity v_n is set on the boundaries where the **Phase Change Interface** node is active, in the same way as if you apply the **Prescribed Normal Mesh Velocity** node. Note that for large deformations of the interface, you may need to add an **Automatic Remeshing** node in the solver sequence to preserve the mesh quality. See Prescribed Normal Mesh Velocity and Automatic Remeshing in the *COMSOL Multiphysics Reference Manual* for details.

PHASE CHANGE INTERFACE

This section defines the parameters used in the Stefan condition.

First, set the **Phase change temperature**, T_{pc} (SI unit: K), and the **Latent heat**, $L_{s \to f}$ (SI unit: J/kg), associated to the transition from solid to fluid phase. A positive value should be set for the latent heat.

Then, choose which side of the phase change interface is the **Solid side**, either **Upside**, or **Downside**.

The red arrow indicates the location of the solid phase, relatively to the selected boundaries in the **Graphics** window. Upside and downside settings are based on 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*.

Finally, choose the method of **Heat flux jump evaluation**. By default the **Lagrange multiplier** option is selected and introduces a variable on the boundary, the Lagrange multiplier, to prescribe the temperature and calculate the heat flux jump accurately.

Alternatively, select **Temperature gradient** to prescribe the temperature condition using a strong constraint and evaluate the heat flux jump from the temperature field gradient on each side of the interface.

The **Temperature gradient** option leads to a formulation that is easier to handle from a numerical point of view, especially for the iterative solvers, however the accuracy of the heat flux jump is strongly dependent on the size of the mesh next to the boundary. In this case a very fine mesh may be needed to reach the same the of accuracy compared to the default option with a default mesh. See Weak Constraints in the *COMSOL Multiphysics Reference Manual* for more details on the use of Lagrange multipliers.

MOVING BOUNDARY SMOOTHING

Numerical oscillations may appear in the interface velocity, with both methods of **Heat flux jump evaluation**.

To overcome this difficulty, select the **Enable moving boundary smoothing** check box to stabilize the boundary displacement. This setting smooths the normal mesh velocity by adding the following smoothing velocity contribution, proportional to the mean curvature of the interface, H (SI unit: $1/m$), and the mesh element size, h (SI unit: m):

$$v_{\mathrm{mbs}} = \delta_{\mathrm{mbs}}|v_{\mathrm{n}}|hH$$

This additional velocity is null for rectilinear interfaces but will artificially increase the normal mesh velocity for highly curved interfaces, such as along droplets or bubbles.

Note that adding this smoothing term modifies the physical modeling, and may alter mass conservation. Set the **Moving boundary smoothing tuning parameter**, δ_{mbs}, a dimensionless number to allow a trade-off between numerical stability and precision of the physical modeling.

See Arbitrary Lagrangian-Eulerian Method.

Tin Melting Front: Application Library path **Heat_Transfer_Module/ Phase_Change/tin_melting_front**

Continuous Casting — Arbitrary Lagrangian-Eulerian Method: Application Library path **Heat_Transfer_Module/Thermal_Processing/ continuous_casting_ale**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Phase Change Interface

Heat Transfer in Solids>Phase Change Interface
Heat Transfer in Fluids>Phase Change Interface
Heat Transfer in Porous Media>Phase Change Interface

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, or **Heat Transfer in Porous Media** selected:

Boundaries>*interface***>Phase Change Interface**

Phase Change Interface, Exterior

Use this node to define the temperature and the position of the interface of a phase change material, using the Stefan condition. It is applicable on exterior boundaries adjacent to Fluid, Solid, and Porous Medium domains, in 2D, 2D axisymmetric, and 3D components.

For an equivalent condition on interior boundaries, use the Phase Change Interface node instead. The settings for these two nodes are similar, except that some properties entering in the expression of the Stefan velocity should be specified on exterior boundaries, depending on which side of the phase change interface the solid phase is located, outside or inside the computational domain. See the Phase Change Interface and Heat Flux sections for details. On interior boundaries the heat flux and solid phase properties are computed from the adjacent domains.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat flux q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The default option for the **Phase Change Interface, Exterior** node is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the boundary on which the node is active.

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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

PHASE CHANGE INTERFACE

This section defines the parameters used in the Stefan condition.

First, set the **Phase change temperature**, T_{pc} (SI unit: K), and the **Latent heat**, $L_{\mathrm{s} \to \mathrm{f}}$ (SI unit: J/kg), associated to the transition from solid to fluid phase. A positive value should be set for the latent heat.

Then, choose which side of the phase change interface is the **Solid side**, either **Inside**, or **Outside** the computational domain.

When the solid phase is outside the computational domain, the **Solid density**, ρ_{s} (SI unit: kg/m^3), and the **Solid translation velocity**, $\mathbf{u}_{\mathrm{s}}$ (SI unit: m/s), should be specified.

When the solid phase is inside the computational domain, the **Fluid density**, ρ_{f} (SI unit: kg/m^3) should be specified.

Both solid and fluid densities are used to evaluate the fluid velocity at the interface, v_{f}, defined as:

$$v_{\mathrm{f}} = \frac{\rho_{\mathrm{f}} - \rho_{\mathrm{s}}}{\rho_{\mathrm{f}}} v_{\mathrm{n}}$$

The red arrow indicates the location of the solid phase, relatively to the selected boundaries in the **Graphics** window.

Finally, choose the method of **Heat flux jump evaluation**. By default the **Lagrange multiplier** option is selected and introduces a variable on the boundary, the Lagrange multiplier, to prescribe the temperature and calculate the heat flux jump accurately.

Alternatively, select **Temperature gradient** to prescribe the temperature condition using a strong constraint and evaluate the heat flux jump from the temperature field gradient at the interface.

The **Temperature gradient** option leads to a formulation that is easier to handle from a numerical point of view, especially for the iterative solvers, however the accuracy of the heat flux jump is strongly dependent on the size of the mesh next to the boundary. In this case a very fine mesh may be needed to reach the same the of accuracy compared to the default option with a default mesh. See Weak Constraints in the *COMSOL Multiphysics Reference Manual* for more details on the use of Lagrange multipliers.

MOVING BOUNDARY SMOOTHING

See Moving Boundary Smoothing in the **Phase Change Interface** node for details about this section.

HEAT FLUX

Click to select the **General inward heat flux** (default), **Convective heat flux**, or **Heat rate** button. Note that the **Convective heat flux** option is only available when the solid phase is inside the computational domain. The settings are the same as in the **Heat Flux** node, see Heat Flux for details.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Phase Change Interface, Exterior
Heat Transfer in Solids>Phase Change Interface, Exterior
Heat Transfer in Fluids>Phase Change Interface, Exterior
Heat Transfer in Porous Media>Phase Change Interface, Exterior

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, or **Heat Transfer in Porous Media** selected:

Boundaries>*interface***>Phase Change Interface, Exterior**

Prescribed Radiosity (Surface-to-Surface Radiation Interface)

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.

COORDINATE SYSTEM SELECTION

Select a boundary system from the **Coordinate system** list to define the reference vector for the azimuthal angle. The first component of the boundary system selected is the reference vector used for directional surface properties. The default is the boundary system of the model, and the list contains any additional boundary system added under the **Definitions** node.

Directional dependence of radiative properties is available when the **Surface-to-surface radiation method** is set to **Ray shooting** in the settings of the Surface-to-Surface Radiation interface.

See Coordinate Systems in the *COMSOL Multiphysics Reference Manual* for more details.

RADIATION DIRECTION

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

RADIOSITY

> **Radiosity** does not directly affect the boundary condition on the boundary where it is specified, but rather defines how that boundary affects others through radiation.

If **Wavelength dependence of radiative properties** is **Constant**, select a **Radiosity expression** between **Graybody radiation** (default), **Blackbody radiation**, or **User defined**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, select a **Radiosity expression** between **Graybody radiation** (default), **Blackbody radiation**, **User defined for each band**, or **User defined**.

Blackbody Radiation
When **Blackbody radiation** is selected it defines the surface radiosity expression corresponding to a blackbody. Define the **External temperature** T_{ext} (SI unit: K, default value 293.15K) to specify the temperature at which the blackbody radiation intensity

e_b is evaluated. Alternatively, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

- When **Wavelength dependence of radiative properties** is set to **Constant** it defines $J = e_b(T_{ext})$ when radiation is defined on one side or $J_u = e_b(T_{ext})$ and $J_d = e_b(T_{ext})$ when radiation is defined on both sides.

- When **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, it defines for each spectral band $J_i = FEP_i(T_{ext})e_b(T_{ext})$ when radiation is defined on one side or $J_{i,d} = FEP_{i,d}(T_{ext})e_b(T_{ext})$ and $J_u = FEP_{i,u}(T_{ext})e_b(T_{ext})$ when radiation is defined on both sides.

	- The blackbody hemispherical total emissive power is defined by $e_b(T) = n^2 \sigma T^4$

Graybody Radiation
When **Graybody radiation** is selected, it sets the surface radiosity expression corresponding to a graybody with a temperature T_{ext}.

Directional dependence of radiative properties is available when the **Surface-to-surface radiation method** is set to **Ray shooting** in the settings of the Surface-to-Surface Radiation interface. By default, the **Directional dependence** of the surface emissivity is set to **Constant**. The **Emissivity** can be defined by an expression depending on other variables such as temperature or spatial coordinates. Switch to the **Polar** or **Polar and azimuthal** options to define the dependences on the polar and azimuthal angles, θ and φ, by specifying a **Directional emissivity function**, f_ε. When using the **Polar** option, select from the **Directional emissivity function** list any function of one argument already defined in the component, under the **Definitions** node. For the **Polar and azimuthal** option, select from the functions of two arguments. By default, only the **Zero** function is available in the list, defining $f_\varepsilon(\theta) = 0$ or $f_\varepsilon(\theta, \varphi) = 0$.

The other dependencies (temperature T, spatial coordinates $\mathbf{x}$) are treated separately in the **Emissivity** expression, and used to define the total emissivity:

$$\varepsilon_{tot}(\theta, \varphi, T, \mathbf{x}) = f_\varepsilon(\theta, \varphi) + \varepsilon(T, \mathbf{x})$$

Select the **Define properties on each side** check box to define specific directional dependences of the surface emissivity on each side of the boundary, and set the **Directional dependence, upside** and **Directional dependence, downside** accordingly.

	The reference vector for the azimuthal angle is given by the first component of the **Coordinate System** selected in the **Coordinate System Selection** section.

By default, the **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 **Emissivity**, enter another value for ε. For **User defined for each band** option, set a value for each spectral band in the table. Within a spectral band, each value is assumed to be independent of wavelength.

If **Wavelength dependence of radiative properties** is set to **Constant**:

- When radiation is defined on one side, define the **Emissivity** ε and the **External temperature** T_{ext} to set $J = \varepsilon e_b(T_{ext})$, or

- When radiation is defined on both sides, define the **Emissivity, upside** ε_u, **Emissivity, downside** ε_d and the **External temperature** T_{ext}. The surface radiosity on upside and downside is then defined by $J_u = \varepsilon_u e_b(T_{ext})$ and $J_d = \varepsilon_d e_b(T_{ext})$ respectively.

If **Wavelength dependence of radiative properties** 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 **Emissivity** ε_i and the **External temperature** T_{ext} to set $J_i = FEP_i \varepsilon_i e_b(T_{ext})$, or

- When radiation is defined on both sides for Bi spectral band, define the **Emissivity, upside** ε_u, **Emissivity, downside** ε_d and the **External temperature** T_{ext}. The surface radiosity on upside and downside is then defined by $J_u = FEP_{Bi}(T_{ext}) \varepsilon_{Bi, u} e_b(T_{ext})$ and $J_d = FEP_{Bi}(T_{ext}) \varepsilon_{Bi, d} e_b(T_{ext})$, 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 radiative properties** 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 value or expression for the **Surface radiosity**, J_0.

Select the **Define prescribed radiosity on each side** check box to set specific values on each side, and define the surface **Radiosity expression** $J_{0,\,u}$ and $J_{0,\,d}$ on the upside and downside, respectively. The geometric normal points from the downside to the upside.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, similar settings are available for each spectral band.

User Defined on Each Band
The surface radiosity $J_{0,i}$ should be set for each spectral band by filling the table. Select the **Define prescribed radiosity on each side** check box to set specific values on each side, and set the **Upside** and **Downside** cells in the table.

🗒	Several settings for this node depend on the **Wavelength dependence of radiative properties** 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

Context Menus
Surface-to-Surface Radiation>Prescribed Radiosity

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries>Prescribed Radiosity

Radiation Group (Surface-to-Surface Radiation Interface)

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 (Surface-to-Surface Radiation Interface), Diffuse Mirror (Surface-to-Surface Radiation Interface), or Prescribed Radiosity (Surface-to-Surface Radiation Interface) 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.

When a boundary 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 (Surface-to-Surface Radiation Interface), Diffuse Mirror (Surface-to-Surface Radiation Interface), and Prescribed Radiosity (Surface-to-Surface Radiation

Interface) 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 (Surface-to-Surface Radiation Interface), Diffuse Mirror (Surface-to-Surface Radiation Interface), and Prescribed Radiosity (Surface-to-Surface Radiation Interface) nodes. The numbers on each boundary specify different groups to which the boundary belongs.
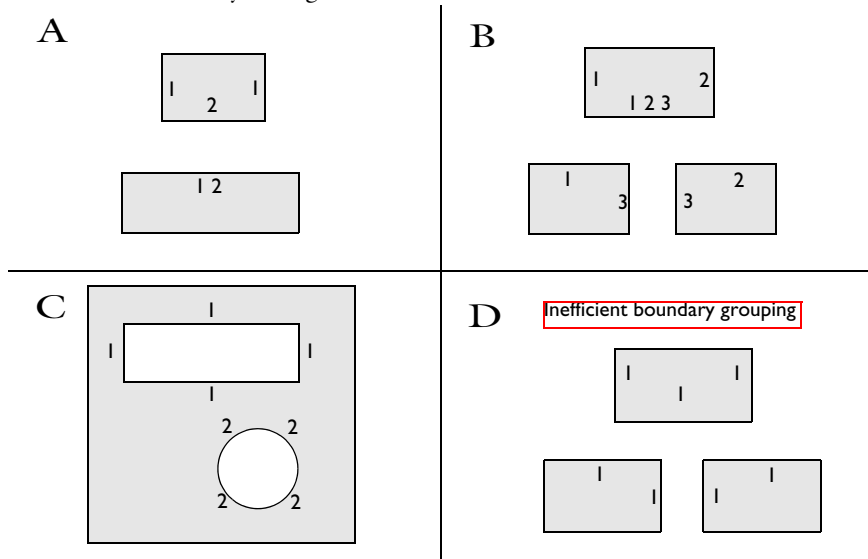
Figure 6-3: Examples of radiation group boundaries.

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 radiative properties** is **Constant**, the radiation group is valid for all wavelengths, and all this section is then empty.

When the **Wavelength dependence of radiative properties** 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 radiative properties** setting, which is defined for the physics interface. See Radiation Settings.

	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:

Surface-to-Surface Radiation>Radiation Group

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries>Radiation Group

Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node defines a boundary semitransparent to radiation. If no radiation is transmitted through the surface, use the Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) node instead.

It prescribes incident intensities on a boundary and takes into account the net radiative heat flux absorbed by the surface, modeling emitted, reflected, and (diffusively and specularly) transmitted radiative intensity on the boundary.

When the **Radiation discretization method** is **Discrete ordinates method**, the net radiative heat flux is defined as the difference between the incoming and outgoing radiative heat fluxes, and the incoming and outgoing radiative heat fluxes are defined from the weighted sums of the incident intensities. See Semitransparent Surface for more details.

When the **Radiation discretization method** is **P1 approximation**, the net radiative heat flux is defined from the incident radiation G. See Semitransparent Surface for more details.

At the internal boundaries, the net radiative heat fluxes, $q_{\mathrm{r,net,d}}$ and $q_{\mathrm{r,net,u}}$ on each side of the surface are defined. Specific radiative properties of the surface can be defined on each side of the boundary.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

MODELS INPUT

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 expression of the blackbody radiative intensity.

> The definition of the boundary temperature may be different from that of the temperature in the adjacent domain.

FRACTIONAL EMISSIVE POWER

This section is available when the **Wavelength dependence of radiative properties** is defined as **Solar and ambient** or **Multiple spectral bands** in the Radiation in Participating Media interface (see Participating Media Settings).

When the **Fractional emissive power** is **Blackbody/Graybody**, the fractional emissive power FEP_k is automatically calculated for each spectral band as a function of the band endpoints and temperature.

When the **Fractional emissive power** is **User defined for each band**, define the **Fractional emissive power**, FEP_k for each spectral band in the table displayed below. All fractional emissive powers are expected to be in $[0,1]$ and their sum must equal 1. Select the **Define fractional emissive power on each side** check box to set specific **Upside** and **Downside** values in the table.

The radiation properties of the boundary and the external radiation intensity should be set in this section.

Emissivity, Diffuse Transmissivity, and Specular Transmissivity

If the **Wavelength dependence of radiative properties** is **Constant**:

- By default, the **Emissivity** ε (dimensionless), **Diffuse transmissivity** τ_d (dimensionless), and **Specular transmissivity** τ_s (dimensionless) use values **From material**. These are properties of the material surface that depend both on the material itself and the structure of the surface. Make sure a material is defined at the boundary level (by default materials are defined at the domain level).

- For **User defined**, set values or expressions. You can define temperature-dependent emissivity and transmissivities using the variable `rad.T`.

Select the **Define properties on each side** check box to set specific values for each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside**, **Emissivity, downside**, **Diffuse transmissivity, upside**, **Diffuse transmissivity, downside**, **Specular transmissivity, upside**, and **Specular transmissivity, downside** are **From material**.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, **Emissivity** ε (dimensionless), **Diffuse transmissivity** τ_d (dimensionless), and **Specular transmissivity** τ_s (dimensionless) use values **From material**.

- When **Emissivity** is **User defined**, enter a value or expression for the **Emissivity** ε. The wavelength is accessible through the `rad.lambda` variable. Any expression set for the emissivity is then averaged on each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Diffuse transmissivity** is **User defined**, enter a value or expression for the **Diffuse transmissivity** τ_d. The wavelength is accessible through the `rad.lambda` variable. Any expression set for the transmissivity is then averaged on each spectral band to obtain a piecewise constant transmissivity. If the average value of the transmissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Specular transmissivity** is **User defined**, enter a value or expression for the **Specular transmissivity** τ_s. The wavelength is accessible through the `rad.lambda`

variable. Any expression set for the specular transmissivity is then averaged on each spectral band to obtain a piecewise constant specular transmissivity. If the average value of the specular transmissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Emissivity** is **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is defined on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

- When **Diffuse transmissivity** is **User defined for each band**, enter a value for the **Diffuse transmissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same transmissivity is defined on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

- When **Specular transmissivity** is **User defined for each band**, enter a value for the **Specular transmissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same transmissivity is defined on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

Set the surface emissivity to a number between 0 and 1, where 0 represents a diffuse mirror and 1 is suitable for a perfect blackbody. The appropriate value for a physical material is somewhere in between and can be found in tables or measurements.

Set the surface diffuse transmissivity to a number between 0 and 1, where 0 applies to an opaque surface and 1 is appropriate for a fully transparent surface.

Set the surface specular transmissivity to a number between 0 and 1, where 0 applies to a perfect diffuse surface and 1 is appropriate when transmissivity is only specular.

The **Specular transmissivity** is only needed when **Discrete ordinates method** is selected.

EXTERNAL RADIATION INTENSITY

The external radiation intensity should be defined when the boundary condition is applied to an external boundary. This intensity is a power per unit solid angle and unit surface area projected onto the plane normal to the direction of radiation. It can be specified for each spectral band.

When the **Wavelength dependence of radiative properties** is set to **Constant**, two options are available to define the external radiation intensity. Set for **External radiation intensity** to **User defined** to define I_{ext} (SI unit: W/m²/sr) directly. Alternatively, select **Blackbody radiation** to define I_{ext} as $I_{\text{b}}(T_{ext})$ and set the **External temperature** T_{ext} (SI unit: K, default value 293.15K) or select an **Ambient temperature** to specify the temperature at which the blackbody radiation intensity I_{b} is evaluated.

When the **Wavelength dependence of radiative properties** is set to **Solar and ambient** or to **Multiple spectral bands**, the external radiation intensity, $I_{\text{ext},k}$ is defined for each spectral band.

Set **External radiation intensity** to **Blackbody radiation** and set the **External temperature** T_{ext} (SI unit: K, default value 293.15K) or select an **Ambient temperature** to define $I_{\text{ext},k}$ from the blackbody intensity $I_{\text{b}}(T_{ext})$, the **External temperature** T_{ext}, and the fractional emissive power for each spectral band at external temperature $FEP_k(T_{ext})$.

Set the **External radiation intensity** to **User defined for each band**, to define $I_{\text{ext},k}$ from user defined expression for each spectral band in the table displayed underneath.

Set the **External radiation intensity** to **User defined distribution**, and define an expression for $I_{\lambda,\text{ext}}$(SI unit: W/m³/sr) to define $I_{\text{ext},k}$ from a spectral distribution. In this case $I_{\text{ext},k}$ is automatically computed per spectral band as the integral of the distribution over each spectral band. If this integral over each band is known, you can use instead the **User defined for each band** option to avoid the numerical overhead due to the evaluation of the average.

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 name of the physics interface node. By default, the Radiation in Participating Media interface is rpm so rpm.sx, rpm.sy, and rpm.sz correspond to the components of discrete ordinate vectors.

- Participating Medium (Radiation in Participating Media Interface)
- Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface)
- Theory for Radiation in Participating Media

Semitransparent Surface (Surface-to-Surface Radiation Interface)

This node is a variant of the Opaque Surface (Surface-to-Surface Radiation Interface) node. In addition to specular and diffuse reflection, specular transmission can also be considered on the surface. It is available when the **Surface-to-surface radiation method** is set to **Ray shooting** in the Surface-to-Surface Radiation interface settings. The node adds a radiosity shape function for each spectral band to its selection and uses it as surface radiosity.

If no radiation is transmitted through the surface, use the Opaque Surface (Surface-to-Surface Radiation Interface) node instead.

Radiation is assumed to occur on both sides of the surface.

MODEL INPUT
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 is used in the expression of 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.

The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be

selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (⬚), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

COORDINATE SYSTEM SELECTION

Select a boundary system from the **Coordinate system** list to define the reference vector for the azimuthal angle. The first component of the boundary system selected is the reference vector used for directional surface properties. The default is the boundary system of the model, and the list contains any additional boundary system added under the **Definitions** node.

See Coordinate Systems in the *COMSOL Multiphysics Reference Manual* for more details.

AMBIENT

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

FRACTIONAL EMISSIVE POWER

These settings are the same as for the Diffuse Surface (Surface-to-Surface Radiation Interface) node.

SURFACE RADIATIVE PROPERTIES

Directional Dependence
By default, the **Directional dependence** of the surface radiative properties (emissivity and transmissivity) is set to **Constant**. The **Emissivity** and **Transmissivity** may be set to expressions depending on other variables like the temperature or the spatial coordinates. Change to **Polar** or **Polar and azimuthal** options to define dependences on the polar and azimuthal angles, θ and φ , through the specification of a **Directional emissivity function**, f_ε and a **Directional transmissivity function**, f_τ . When the **Polar** option is used, select in the list any function of one argument already defined in the component, under the **Definitions** node. For the **Polar and azimuthal** option, select among the functions of two arguments. By default, only the **Zero** function is available in the list, defining $f_\varepsilon(\theta) = 0$, $f_\tau(\theta) = 0$, $f_\varepsilon(\theta, \varphi) = 0$, or $f_\tau(\theta, \varphi) = 0$.

The other dependencies (temperature T, spatial coordinates **x**) are handled separately in the **Emissivity** and **Transmissivity** expressions, and used to define the total properties:

$$\varepsilon_{tot}(\theta, \varphi, T, \mathbf{x}) = f_\varepsilon(\theta, \varphi) + \varepsilon(T, \mathbf{x})$$

$$\tau_{tot}(\theta, \varphi, T, \mathbf{x}) = f_\tau(\theta, \varphi) + \tau(T, \mathbf{x})$$

Emissivity, Diffuse Reflectivity, Transmissivity and Critical Angle

If **Wavelength dependence of radiative properties** is **Constant**:

- By default, the **Emissivity** ε (dimensionless), **Diffuse reflectivity** ρ_d (dimensionless), and **Transmissivity** τ (dimensionless) use values **From material**. These are properties of the material surface that depend 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).

- For **User defined**, set values or expressions. You may set temperature-dependent emissivity and reflectivity using the variable rad.T.

Select the **Define properties on each side** check box to set specific values on each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside**, **Emissivity, downside**, **Diffuse reflectivity, upside**, **Diffuse reflectivity, downside**, **Transmissivity, upside**, **Transmissivity, downside** are **From material**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Emissivity** ε (dimensionless), **Diffuse reflectivity** ρ_d (dimensionless), and **Transmissivity** τ (dimensionless) use values **From material**.

- When **Emissivity** is set to **User defined**, enter a value or expression for the **Emissivity** ε. The wavelength may be accessed through the rad.lambda variable. Any expression set for the emissivity is then averaged on each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Diffuse reflectivity** is set to **User defined**, enter a value or expression for the **Diffuse reflectivity** ρ_d. The wavelength may be accessed through the rad.lambda variable. Any expression set for the reflectivity is then averaged on each spectral band to obtain a piecewise constant reflectivity. If the average value of the reflectivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Transmissivity** is set to **User defined**, enter a value or expression for the **Transmissivity** τ. The wavelength may be accessed through the rad.lambda variable.

Any expression set for the transmissivity is then averaged on each spectral band to obtain a piecewise constant transmissivity. If the average value of the transmissivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Emissivity** is set to **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is defined on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

- When **Diffuse reflectivity** is set to **User defined for each band**, enter a value for the **Diffuse reflectivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same reflectivity is defined on both sides. Select the **Define diffuse reflectivity on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

- When **Transmissivity** is set to **User defined for each band**, enter a value for the **Transmissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same transmissivity is defined on both sides. Select the **Define properties on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

Set the surface emissivity to a number between 0 and 1, where 0 represents a 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.

Set the surface diffuse reflectivity to a number between 0 and 1, where 1 applies to a perfect diffuse surface and 0 is appropriate when reflection is only specular.

Set the surface transmissivity to a number between 0 and 1, where 0 applies to an opaque surface and 1 is appropriate for a fully transparent surface.

The surface specular reflectivity, ρ_s, is then defined as

$$\rho_s = 1 - \rho_d - \varepsilon - \tau$$

Set a value or expression for the **Critical angle** θ_c, which is the minimum angle of incidence (measured between the ray and the normal to the surface) for transmission to occur. Below this angle, all the radiation is reflected. The `pi/2` default value ensures that total reflection does not occur by default.

When the **Radiating side** is set to **None** for a spectral band, the information set for this spectral band in this section is not used.

Surface-to-Ambient Radiation (Heat Transfer Interface)

Use this node to take into account radiation from boundaries to the ambient. The net inward heat flux from surface-to-ambient radiation is

$$-\mathbf{n} \cdot \mathbf{q} = \varepsilon\sigma(T_{\mathrm{amb}}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

MODEL INPUT
This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature
This section is available when material properties are temperature-dependent. 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), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Define on which side of the boundary is the ambient domain that receives the radiation, by selecting a **Radiating side**: **Upside** or **Downside**.

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*.

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 an **Ambient Properties** node under **Definitions**.

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.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids>Heat Sources>Surface-to-Ambient Radiation
Heat Transfer in Porous Media>Porous Medium>Fluid>Heat Sources>Surface-to-Ambient Radiation
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Heat Sources>Surface-to-Ambient Radiation

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids**, or any version of the Heat Transfer interface selected:

Boundaries>Surface-to-Ambient Radiation

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Surface-to-Ambient Radiation

Symmetry (Heat Transfer Interface)

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 take into account 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 in Solids and Fluids>Symmetry
Heat Transfer in Porous Media>Porous Medium>Fluid>Symmetry
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Symmetry

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Symmetry**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Symmetry

Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node provides a condition for symmetry boundaries when the **Radiation discretization method** is **P1 approximation**. It means that there is no heat flux across the boundary.

LOCATION IN USER INTERFACE

Context Menus
Radiation in Participating Media>Symmetry
Radiation in Absorbing-Scattering Media>Symmetry

Ribbon
Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Boundaries>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 to apply this condition to. A pair must 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 an **Ambient Properties** node under **Definitions**.

CONSTRAINT SETTINGS
To display this section, click the **Show More Options** 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 constraints** are not supported for resistive thin layers or with turbulent wall functions.

You can specify a harmonic variation of the temperature value in a frequency domain analysis of perturbation type, by adding a **Harmonic Perturbation** subnode. See Theory for Harmonic Heat Transfer for more details.

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 in Solids and Fluids>Temperature
Heat Transfer in Porous Media>Porous Medium>Fluid>Temperature
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Temperature

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Temperature**
Pairs>*interface***>Temperature**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>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 **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 version of **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 to apply this condition to. A pair must to be created first. See Identity and Contact Pairs in the *COMSOL Multiphysics Reference Manual* for more details.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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** (![icon]). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (![icon]), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Select an option in the **Material type** list to specify if the inputs of the **Thermal Friction** section are defined in the material or spatial frame:

- The default option for the **Thermal Contact** node is **Solid**, which specifies that the heat source Q_b is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_b in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the boundary on which the node is active.

📑   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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

THERMAL CONTACT

Select a **Contact model**: **Constriction conductance with interstitial gas** (default), or **Equivalent thin resistive layer**.

With the **Constriction conductance with interstitial gas** model, you consider the microscopic configuration of the joint, by setting both the constriction conductance at the contact spots, h_c, and the gap conductance due to the fluid in the interstitial space, h_g, to evaluate the joint conductance. The characteristic size of the microscopic surface asperities, and both the conductance of constriction and gap need to be specified.

To represent the surfaces asperities and the fluid gap between these surfaces by an equivalent thin layer, select the **Equivalent thin resistive layer** option instead.

The radiative conductance can be accounted for with the two contact models. It should be considered at high temperatures, above 600°C.

See Theory for Thermal Contact for details.

Depending on the selected **Contact model**, further settings display underneath.

Constriction Conductance with Interstitial Gas

Select a **Constriction conductance**: **Cooper-Mikic-Yovanovich correlation** (default), **Mikic elastic correlation**, or **User defined**. For **User defined** enter a value or expression for h_c.

Then select the **Gap conductance**: **User defined** (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.

Equivalent Thin Resistive Layer

The available options to specify the resistive behavior of the equivalent layer are **Layer resistance** (default), **Layer conductance**, and **Layer thermal conductivity and thickness**. Depending on the selected option, enter values or expressions for the **Layer resistance**, R_{eq}, the **Layer conductance**, h_{eq}, the **Layer thermal conductivity**, k_{eq}, and the **Layer thickness**, d_s.

Radiative Conductance

Finally, choose the **Radiative conductance**: **User defined** (default) or **Gray-diffuse parallel surfaces**. For **User defined** enter a value for h_r.

CONTACT SURFACE PROPERTIES

This section is available when the **Contact model** is **Constriction conductance with interstitial gas**, if **Cooper-Mikic-Yovanovich correlation** or **Mikic elastic correlation** are chosen as the **Constriction conductance correlation** for the **Constriction conductance**. 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** (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 a **Contact interface Young's modulus** E_{contact}: **Weighted harmonic mean** (default) or **User defined**.

- For **Weighted harmonic mean**, enter values or expressions for the **Young's modulus, upside**, E_{u}, the **Young's modulus, downside**, E_{d}, the **Poisson's ratio, upside**, ν_{u}, and the **Poisson's ratio, downside**, ν_{d}. If this node is selected from the **Pairs** menu, enter instead values or expressions for the **Young's modulus, source**, E_{src}, the **Young's modulus, destination**, E_{dst}, the **Poisson's ratio, source**, ν_{src}, and the **Poisson's ratio, destination**, ν_{dst}.

- For **User defined** enter another value or expression for E_{contact}.

GAP PROPERTIES

This section is available when the **Contact model** is **Constriction conductance with interstitial gas**, if **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 **Full** 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** (default) or **User defined**. For **User defined** enter a value for r.

Select either the **General source** (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.

In addition, surface plots for the temperatures on the upside (ht.Tu) and downside (ht.Td) of the slit are automatically generated in 3D components.

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 in Solids and Fluids>Thin Structures>Thermal Contact

Heat Transfer in Solids>Thin Structures>Thermal Contact

Heat Transfer in Fluids>Thin Structures>Thermal Contact

Heat Transfer in Porous Media>Thin Structures>Thermal Contact

Bioheat Transfer>Thin Structures>Thermal Contact

Heat Transfer in Solids>Pairs>Thermal Contact

Heat Transfer in Fluids>Pairs>Thermal Contact

Heat Transfer in Porous Media>Pairs>Thermal Contact

Bioheat Transfer>Pairs>Thermal Contact

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials** or **Bioheat Transfer** selected:

Pairs>*interface***>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. As the default boundary condition, it can be applied only on external boundaries; but when added manually, it can be applied also on interior boundaries.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids and Fluids>Thermal Insulation
Heat Transfer in Porous Media>Porous Medium>Fluid>Thermal Insulation
Heat Transfer in Porous Media>Porous Medium>Porous Matrix>Thermal Insulation

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries>*interface***>Thermal Insulation**

Physics tab with **Porous Medium>Fluid** or **Porous Medium>Porous Matrix** selected in the model tree:

Attributes>Thermal Insulation

Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)

This node behaves like Fluid but is applicable on boundaries.

BOUNDARY SELECTION

When the **Fluid** node is added manually in the Heat Transfer in Films interface, and for the **Thin Film** node, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

When **Fluid** is the default node of the Heat Transfer in Films interface, it is applied on all boundaries where the Heat Transfer in Films interface is applied, and neither the boundary selection nor the **Restrict to layered boundaries** check box are editable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Restrict to layered boundaries** check box is not editable.

SHELL PROPERTIES

Fluid (Heat Transfer in Shells Interface)
Different settings are available, depending on the settings in the Shell Properties section of the parent interface, and whether **Fluid** is the default node or was added manually:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Fluid** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers. In this case, both the **General** and **Thermally thin approximation** options are available in the **Layer Model** section.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is taken **From physics interface** in the **Fluid** node. This option is not editable when **Fluid** is the default node, but you can change to **User defined** and override the interface setting with a specific value or expression for L_{th} when the node was added manually. In this case, only the **Thermally thin approximation** option is available in the **Layer Model** section.

Thin Film (Heat Transfer Interface)

Two options are available for the **Shell type**:

- When the **Shell type** is **Layered shell**, the Extra Dimension tool is used to solve the equations through the thickness of a layered material. It is possible to consider several layers with different thermal properties varying through the thickness. This makes the **General** and **Thermally thin approximation** options available for **Layer type** in the **Layer Model** section. You can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- Alternatively, set **Shell type** to **Nonlayered shell**, and set a user defined value or expression for the **Thickness** L_{th}. This option should be used for thermally thin layers, for which no through-thickness temperature variation is expected in the layered material. This makes **Thermally thin approximation** the only option available for **Layer type** in the **Layer Model** section.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

⊕	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

⊕	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include 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. The **Common model input** option corresponds to the `minput.pA` variable, set to 1 atm by default. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section.

LAYER MODEL

The available options for **Layer type** are **Thermally thin approximation**, and **General** (when the **Shell type** is **Layered material** in the Shell Properties section). The former is a lumped model that accounts only for tangential temperature gradients (along the film), whereas the latter accounts 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.

If the **Layer type** is **Thermally thin approximation**, and the **Shell type** is **Layered shell**, the **Layerwise constant properties** check box is available to solve the equations with weighted averaged material properties, assuming that these properties are constant within each layer. When suitable, this allows to compute efficiently the homogenized properties of the layered material. By default this check box is not selected, and the properties are integrated through the thickness of the layered material. It means that you can provide material parameters with a through-thickness variation by using

expressions containing the extra dimension coordinate, as described in Using the Extra Dimension Coordinates.

HEAT CONVECTION

The settings are the same as in the Fluid domain feature.

HEAT CONDUCTION, FLUID

The settings are the same as in the Fluid domain feature.

THERMODYNAMICS, FLUID

The settings are the same as in the Fluid domain feature, except for the available options in the **Fluid type** list: **Ideal gas**, **Gas/Liquid**, and **Moist air**.

See the Fluid domain feature for more details on the **Ideal gas** and **Gas/Liquid** options.

Moist Air

If **Moist air** is selected, the thermodynamics properties are defined as a function of the amount of vapor in moist air. The **Input quantity** options available to define this amount are as follows:

- **Vapor mass fraction** to define the ratio between vapor mass and 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 to the **Model Inputs** section.

- **Moisture content** (default), also called mixing ratio or humidity ratio, to define the ratio between water vapor mass and dry air mass. For **User defined**, enter a value or expression for the **Moisture Content** x_{vap}. Else, select an **Ambient moisture content** defined in an **Ambient Properties** node under **Definitions**.

- **Relative humidity** ϕ_w, a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to air saturated with water vapor. The **Relative humidity, temperature condition** T_{ϕ_w} and **Relative humidity, absolute pressure condition** p_{ϕ_w} should be specified.

Ⓠ   Within a layered material selection, a single **Layer type** should be used. If two layer types are needed for the same layered material, the original material should be duplicated so that one layered material is defined for each layer type. A Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface) node may be added between the two layered materials.

> ⚠️ The thermal conductivity of the thin film is assumed to be isotropic, even if an anisotropic property is defined for the material.

Context Menus
Heat Transfer in Fluids>Thin Structures>Thin Film
Heat Transfer in Films>Fluid

Ribbon
Physics tab with *interface* as **Heat Transfer in Fluids** selected:

Boundaries>Thin Film

Physics tab with *interface* as **Heat Transfer in Films** selected:

Boundaries>Fluid

Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)

These nodes define the thermal conductivity and thermodynamics properties of a material located on internal or external boundaries. The **Thin Layer** node is available under the Heat Transfer interface, while the **Solid** node is available under the Heat Transfer in Shells interface.

The 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 thermal properties. Each sub-layer 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 sub-layers 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 thermal properties, and heat sources can be applied on a sub-layer selection, and on up and down sides of the layer.

BOUNDARY SELECTION

When the **Solid** node is added manually in the Heat Transfer in Shells interface, and for the **Thin Layer** node, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

When **Solid** is the default node of the Heat Transfer in Shells interface, it is applied on all boundaries where the Heat Transfer in Shells interface is applied, and neither the boundary selection nor the **Restrict to layered boundaries** check box are editable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Restrict to layered boundaries** check box is not editable.

SHELL PROPERTIES

Solid (Heat Transfer in Shells Interface)
Different settings are available, depending on the settings in the Shell Properties section of the parent interface, and whether **Solid** is the default node or was added manually:

• When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Solid** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box, when the node was added manually. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers. In this case, both the **General** and **Thermally thin approximation** options are available in the **Layer Model** section.

• When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is taken **From physics interface** in the **Solid** node. This option is not editable when **Solid** is the default node, but you can change to **User defined** and override the interface's setting with a specific value or expression for L_{th} when

the node was added manually. In this case, only the **Thermally thin approximation** option is available in the **Layer Model** section.

Thin Layer (Heat Transfer Interface)
Two options are available for the **Shell type**:

- When the **Shell type** is **Layered shell**, the Extra Dimension tool is used to solve the equations through the thickness of a layered material. It is possible to consider several layers with different thermal properties varying through the thickness. This makes the **General**, **Thermally thin approximation**, and **Thermally thick approximation** options available for **Layer type** in the **Layer Model** section. You can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- Alternatively, set **Shell type** to **Nonlayered shell**, and set a user defined value or expression for the **Thickness** L_{th}. This option should be used for thermally thin layers, for which no through-thickness temperature variation is expected in the layered material. This removes the **General** option from the **Layer type** list in the **Layer Model** section.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

⚲	- For a general description of layer and interface selections, see Layer and Interface Selection Tools. - You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

⚲	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable minput.Tempref, which is set to 293.15 K by default. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

⚠ This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

LAYER MODEL

The available options for **Layer type** are **Thermally thin approximation**, **Thermally thick approximation** (available only in the **Thin Layer** feature), and **General**.

If **Layer type** is **Thermally thick approximation**, from the **Specify** list select **Layer properties** (default) or **Thermal resistance**.

If the **Layer type** is **Thermally thin approximation**, and the **Shell type** is **Layered shell**, the **Layerwise constant properties** check box is available to solve the equations with weighted averaged material properties, assuming that these properties are constant within each layer. When suitable, this allows to compute efficiently the homogenized properties of the layered material. By default this check box is not selected, and the properties are integrated through the thickness of the layered material. It means that you can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate, as described in Using the Extra Dimension Coordinates.

> 🔍 Within a layered material selection, a single **Layer type** should be used. If two layer types are needed for the same layered material, the original material should be duplicated so that one layered material is defined for each layer type. A Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface) node may be added between the two layered materials.

HEAT CONDUCTION

The default **Thermal conductivity** k is taken **From layered material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** to enter another value or expression.

Alternatively, set a value for the **Thermal resistance** R_s if **Specify** is set to **Thermal resistance** in the **Layer Model** section (for **Thermally thick approximation**).

THERMODYNAMICS

By default the **Density** ρ and **Heat capacity at constant pressure** C_p of the layer are taken **From layered material**. See Material Density in Features Defined in the Material Frame if a temperature-dependent density should be set. For **User defined** enter other values or expressions.

When **Layer type** is **Thermally thick approximation**, these properties are only used in time-dependent studies, but must be set in all cases.

These subnodes are available for **Thin Layer** node:

- Heat Source (Thin Layer, Thin Film, Fracture) — to add a layer internal heat source, Q_s, within the layer.
- 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 Shells Interface) — to set a prescribed temperature condition on a specified set of boundaries.
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface) — to add a surface-to-ambient radiation for the layer end.

When multiple layers are defined they are numbered from the downside (Layer 1) to the upside. 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*.

- Theory for Heat Transfer in Thin Structures
- Boundary Wall Temperature
- Plotting and Evaluating Results in Layered Materials

- *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**

Context Menus
Heat Transfer in Solids>Thin Structures>Thin Layer
Heat Transfer in Shells>Solid

Ribbon
Physics tab with *interface* as **Heat Transfer in Solids** selected:

Boundaries>Thin Layer

Physics tab with *interface* as **Heat Transfer in Shells** selected:

Boundaries>Solid

Transparent Surface (Radiative Beam in Absorbing Medium Interface)

Use this node to model an absorbing medium's boundary where there is no radiative intensity absorption. All the outgoing beams traveling from the medium to the exterior leave the media without depositing any energy at this boundary. The intensity of radiative beams entering the absorbing media is set to zero. See Opaque Surface (Radiative Beam in Absorbing Medium Interface) to take into account radiative intensity absorption on boundaries.

> 🔍 Radiative Beam in Absorbing Media Theory

LOCATION IN USER INTERFACE

Context Menus
Radiative Beam in Absorbing Media>Transparent Surface

Ribbon
Physics tab with **Radiative Beam in Absorbing Media** selected:

Boundaries>Transparent Surface

Boundary Interface Features

The Heat Transfer interfaces have the following boundary interface nodes and subnodes available:

- Deposited Beam Power, Interface (Heat Transfer in Shells Interface)
- Heat Flux, Interface (Heat Transfer in Shells Interface)
- Heat Source, Interface (Heat Transfer in Shells Interface)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)

- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in Shells Interface)
- Thermal Insulation, Interface (Heat Transfer in Shells Interface)

	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

Deposited Beam Power, Interface (Heat Transfer in Shells Interface)

This node models heat sources brought by narrow beams, such as laser or electron beams, to the exterior interfaces of a shell and, for the case of a layered shell, to the interfaces between its layers.

BOUNDARY SELECTION

Select the boundaries on which to apply the heat source.

In addition, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

Set the interfaces for which the heat source contribution should be applied. These can be internal interfaces between layers or the top and bottom surfaces.

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See Interface Selections for more details on the interfaces identification.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**. This setting has no effect unless the temperature differs from one side of the boundary to the other.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

Q	- For a general description of layer and interface selections, see Layer and Interface Selection Tools. - You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

Q	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

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.*

BEAM ORIENTATION

Enter a value for the **Beam orientation e**. This vector does not need to be normalized. In 2D axisymmetric components, it is aligned with the z-axis.

BEAM PROFILE

Either select an option for the **Beam profile** among the **Built-in beam profiles**, or set the it as **User defined** by entering a value for the **Beam power density**, I_0 (SI unit: W/m^2). The **Beam power density** corresponds to the beam intensity in the beam cross section.

For **Built-in beam profiles**, enter a value for the **Deposited beam power** P_0 and the coordinates of the **Beam origin point O**.

Then, select a **Distribution type**: **Gaussian** (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.

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.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Shells>Interfaces>Deposited Beam Power, Interface
Heat Transfer in Films>Interfaces>Deposited Beam Power, Interface

Heat Transfer in Fractures>Interfaces>Deposited Beam Power, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Deposited Beam Power, Interface

Heat Flux, Interface (Heat Transfer in Shells Interface)

This node adds a heat flux q_0 on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers.

BOUNDARY SELECTION

Select the boundaries on which to apply the heat flux.

In addition, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

INTERFACE SELECTION

Set the interfaces for which the heat flux should be applied. These can be internal interfaces between layers or the top and bottom surfaces.

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See Interface Selections for more details on the interfaces identification.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**. This

setting has no effect unless the temperature differs from one side of the boundary to the other.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

Q
- For a general description of layer and interface selections, see Layer and Interface Selection Tools.
- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

Q
The desired selection for the node may correspond to boundaries with different layered materials.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

Q
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*.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat fluxes q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The default option for the **Heat Flux** node is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

📄	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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

HEAT FLUX

These settings are the same as for the Heat Flux node.

📄	In 2D the heat flux contribution is multiplied by d_z to take into account the out-of-plane thickness.

🔍	• Theory for Heat Transfer in Thin Structures • Convective Heat Transfer Correlations

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells>Interfaces>Heat Flux, Interface
Heat Transfer in Films>Interfaces>Heat Flux, Interface
Heat Transfer in Fractures>Interfaces>Heat Flux, Interface

Ribbon
Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Heat Flux, Interface

Heat Source, Interface (Heat Transfer in Shells Interface)

This node models a heat source (or sink) on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers. It adds a heat source $q = Q_b$ or $q = P_b/A$. A positive q is heating and a negative q is cooling.

BOUNDARY SELECTION

Select the boundaries on which to apply the heat source.

In addition, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

INTERFACE SELECTION

Set the interfaces for which the heat source contribution should be applied. These can be internal interfaces between layers or the top and bottom surfaces.

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

• When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See Interface Selections for more details on the interfaces identification.

• When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**. This setting has no effect unless the temperature differs from one side of the boundary to the other.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

> - For a general description of layer and interface selections, see Layer and Interface Selection Tools.
> - You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

> The desired selection for the node may correspond to boundaries with different layered materials.
>
> See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

> 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*.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Boundary Heat Source** section are defined in the material or spatial frame:

- The default option for the **Heat Source** node is **Solid**, which specifies that the heat source Q_b is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_b in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

	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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

BOUNDARY HEAT SOURCE

Click the **General source** (default) or **Heat rate** button.

For **General source**, 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 $Q_b = P_b/A$, where A is the total area of the selected layers interface.

	In 2D components, the equation contains an additional factor, d_z, to take into account the out-of-plane thickness. This is because the selected points correspond to edges in a 3D geometry.

	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

	• Theory for Heat Transfer in Thin Structures • Handling Frames in Heat Transfer

Context Menus

Heat Transfer in Shells>Interfaces>Heat Source, Interface
Heat Transfer in Films>Interfaces>Heat Source, Interface
Heat Transfer in Fractures>Interfaces>Heat Source, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Heat Source, Interface

Lumped System Connector, Interface (Heat Transfer in Shells Interface)

Use this node to connect the distributed finite element model to a thermal network.

The average temperature on the selected interface can be prescribed at a node of the thermal network where an External Terminal feature of The Lumped Thermal System Interface is applied.

In return, the heat rate defined at the node of the thermal network is applied on the selected interface.

BOUNDARY SELECTION

Select the boundaries on which to apply the connector.

In addition, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

INTERFACE SELECTION

Set the interfaces for which the connector should be applied. These can be the top and bottom surfaces of the layered material.

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, and **Selected interfaces**. With the last option you can select the interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See Interface Selections for more details on the interfaces identification.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

Q	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

Q	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

Q	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*.

Select an **External Terminal** feature in the **Source** list to define which node of the thermal network provides the heat rate P_{ext} to be prescribed on the interface.

When the **Isothermal terminal** check box is not selected (default), the average temperature on the interface is constrained to the temperature at the terminal node of the thermal network. Select this check box to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.

▥	*Lumped Composite Thermal Barrier with Shells*: Application Library path **Heat_Transfer_Module/Tutorials,_Thin_Structure/ lumped_composite_thermal_barrier_shells**

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Shells>Interfaces>Lumped System Connector, Interface
Heat Transfer in Films>Interfaces>Lumped System Connector, Interface
Heat Transfer in Fractures>Interfaces>Lumped System Connector, Interface

Ribbon
Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Lumped System Connector, Interface

Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)

Use this node to add surface-to-ambient radiation on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers. The net inward heat flux due to surface-to-ambient radiation is

$$-\mathbf{n} \cdot \mathbf{q} = \varepsilon\sigma(T_{\text{amb}}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

BOUNDARY SELECTION
Select the boundaries on which to add surface-to-ambient radiation.

In addition, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

INTERFACE SELECTION

Set the interfaces for which surface-to-surface radiation should be considered. These can be internal interfaces between layers or the top and bottom surfaces.

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See Interface Selections for more details on the interfaces identification.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**. This setting has no effect unless the temperature differs from one side of the boundary to the other.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

| Q | - For a general description of layer and interface selections, see Layer and Interface Selection Tools.
- You can provide material parameters with a through-thickness variation by explicitly or implicitly using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates. |

The desired selection for the node may correspond to boundaries with different layered materials.

Q̇   See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

Q̇   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*.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

SURFACE-TO-AMBIENT RADIATION

Surface Emissivity

The default **Surface emissivity** ε (a dimensionless number between 0 and 1) is taken **From layered 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}. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

🗒	In 2D, the equation has an additional factor, d_z, to take into account the out-of-plane thickness.

🔍	Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells>Interfaces>Surface-to-Ambient Radiation, Interface
Heat Transfer in Films>Interfaces>Surface-to-Ambient Radiation, Interface
Heat Transfer in Fractures>Interfaces>Surface-to-Ambient Radiation, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Surface-to-Ambient Radiation, Interface

Temperature, Interface (Heat Transfer in Shells Interface)

Use this node to specify the temperature on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers.

BOUNDARY SELECTION

Select the boundaries on which to specify the temperature.

In addition, select the **Restrict to layered boundaries** check box to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

This section is available only when the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent physics interface.

Set the interfaces for which the temperature should be prescribed. These can be internal interfaces between layers or the top and bottom surfaces.

The available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See Interface Selections for more details on the interfaces identification.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

	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*.

For **User defined**, enter a value or expression for the **Temperature** T_0. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

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 More Options** button (👁) and select **Advanced Physics Options**.

📋 | You can specify a harmonic variation of the temperature value in a frequency domain analysis of perturbation type, by adding a **Harmonic Perturbation** subnode. See Theory for Harmonic Heat Transfer for more details.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Shells>Interfaces>Temperature, Interface
Heat Transfer in Films>Interfaces>Temperature, Interface
Heat Transfer in Fractures>Interfaces>Temperature, Interface

Ribbon
Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Temperature, Interface

Thermal Contact, Interface (Heat Transfer in Shells Interface)

Use this node to specify thermal contact at the internal interfaces between the layers of a layered shell. The surfaces asperities of the interfaces and the fluid gap between these surfaces are represented by an equivalent thin layer, for which the thermal conductance should be specified. This node is available only when the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent physics interface.

BOUNDARY SELECTION

Select the boundaries on which to specify the thermal contact. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

Set the interfaces for which the thermal contact should be prescribed. These can be only internal interfaces between layers.

The available options in the **Apply to** list are **Interior interfaces** and **Selected interfaces**. With the second option you can select any set of internal interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. See Interface Selections for more details on the interfaces identification.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

Q | • For a general description of layer and interface selections, see Layer and Interface Selection Tools.
• You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

Q | The desired selection for the node may correspond to boundaries with different layered materials.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

Q | 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*.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

THERMAL CONTACT

The available options to specify the resistive behavior of the gap are **Gap resistance** (default), **Gap conductance**, and **Gap thermal conductivity and thickness**. Depending on the selected option, enter values or expressions for the **Gap resistance**, R_{eq}, the **Gap conductance**, h_{eq}, the **Gap thermal conductivity**, k_{eq}, and the **Gap thickness**, d_s.

Radiative Conductance

The radiative conductance should be considered at high temperatures, above 600°C. Choose the **Radiative conductance**: **User defined** (default) or **Gray-diffuse parallel surfaces**. For **User defined** enter a value for h_r.

RADIATIVE CONDUCTANCE

This section is available when **Gray-diffuse parallel surfaces** is selected as the **Radiative conductance** under **Thermal Contact**.

By default the **Surface emissivity, upside** and the **Surface emissivity, downside**, ε_u and ε_d, are taken **From material**. For **User defined** enter another value or expression.

🔍	Theory for Thermal Contact

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells>Interfaces>Thermal Contact, Interface
Heat Transfer in Films>Interfaces>Thermal Contact, Interface
Heat Transfer in Fractures>Interfaces>Thermal Contact, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Thermal Contact, Interface

Thermal Insulation, Interface (Heat Transfer in Shells Interface)

Use this node to specify thermal insulation on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers.

BOUNDARY SELECTION

Select the boundaries on which to specify the thermal insulation. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, slmat1), otherwise the boundary is marked as not applicable.

INTERFACE SELECTION

Set the interfaces for which the thermal insulation should be prescribed. These can be internal interfaces between layers or the top and bottom surfaces.

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See Interface Selections for more details on the interfaces identification.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

🔍
- For a general description of layer and interface selections, see Layer and Interface Selection Tools.
- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

🔍
The desired selection for the node may correspond to boundaries with different layered materials.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

🔍
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 in Shells>Interfaces>Thermal Insulation, Interface
Heat Transfer in Films>Interfaces>Thermal Insulation, Interface
Heat Transfer in Fractures>Interfaces>Thermal Insulation, Interface

Ribbon
Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries>Thermal Insulation, Interface

Edge Features

The Heat Transfer interfaces have the following edge nodes and subnodes available:

- Heat Flux (Heat Transfer in Shells Interface)
- Heat Flux (Thin Layer, Thin Film, Fracture)
- Heat Source (Heat Transfer in Shells Interface)
- Line Heat Source
- Lumped System Connector (Heat Transfer in Shells Interface)
- Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)

- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thin Rod

🔍 For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

Heat Flux (Heat Transfer in Shells Interface)

Use this node to add heat flux across boundaries of a 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 $q = q_0$.

EDGE SELECTION

Select the edges on which to apply the heat flux. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, `slmat1`), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Heat Flux** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thikness** is **From physics interface** in the **Heat Flux** node. You can change to **User defined** and override the interface setting with a specific value or expression for L_{th}.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

🔍	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

🔍	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat flux q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The default option for the **Heat Flux** node is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

📝	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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

HEAT FLUX

These settings are the same as for the Heat Flux (Thin Layer, Thin Film, Fracture) node available for the other interfaces.

📝	In 2D, $q = d_z q_0$ to take into account 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**

Context Menus
Heat Transfer in Shells>Heat Flux

Ribbon
Physics tab with **Heat Transfer in Shells** selected:

Egdes>Heat Flux

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.

EDGE SELECTION

Select the edges on which to apply the heat flux. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, slmat1), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent node, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent node:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent node, the same layered material is used in the **Heat Flux** node. This option is not editable, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is taken **From parent feature** in the **Heat Flux** node. You can change to **User defined** and override the parent node setting with a specific value or expression for L_{th}.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

Ⓠ
- For a general description of layer and interface selections, see Layer and Interface Selection Tools.
- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

Ⓠ
The desired selection for the node may correspond to boundaries with different layered materials.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat flux q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The default option for the **Heat Flux** subnode is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

HEAT FLUX

The options available in the **Flux type** list are: **General inward heat flux** (default), **Convective heat flux**, **Nucleate boiling heat flux**, and **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 **Convective heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{ext} - T)$. 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 an **Ambient Properties** node under **Definitions**. This option is only available when the **Layer type** is **Thermally thin approximation** or **General** in the parent feature.

- If **Nucleate boiling heat flux** is selected, it adds q_0 estimated with the Rohsenow's correlation. See Nucleate Boiling Heat Flux for the settings. This option is only

available when the **Layer type** is **Thermally thin approximation** or **General** in the parent feature.

- *3D Components*: If **Heat rate** is selected, it adds q_0 in the form $q_0 = P_0/A$ where A is equal to the area of the edge selection. Enter the heat rate P_0.

ⓠ
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
- Theory for Heat Transfer in Thin Structures
- Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Solids>Thin Layer>Heat Flux
Heat Transfer in Solids>Thin Film>Heat Flux
Heat Transfer in Porous Media>Fracture>Heat Flux

Ribbon
Physics tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:

Attributes>Heat Flux

Heat Source (Heat Transfer in Shells Interface)

This node models a linear heat source (or sink). It adds a heat source $q = Q_b$ or $q = P_b/A$. A positive q is heating and a negative q is cooling.

EDGE SELECTION
Select the edges on which to apply the heat source. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, `slmat1`), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Heat Source** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is taken **From physics interface** in the **Heat Source** node. You can change to **User defined** and override the interface setting with a specific value or expression for L_{th}.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	- For a general description of layer and interface selections, see Layer and Interface Selection Tools.
Q	- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

	The desired selection for the node may correspond to boundaries with different layered materials.
Q	See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Source** section are defined in the material or spatial frame:

- The default option for the **Heat Source** node is **Solid**, which specifies that the heat source Q_b is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_b in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

📄	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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

BOUNDARY HEAT SOURCE

Select an option from the **Heat Source** list:

For **General source** (default), 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 $Q_b = P_b/A$, where A is the total area of the selected edges (where A is the product of the layer's thickness d_s by the total length of the selected edges).

📄	In 2D components, the equation contains an additional factor, d_z, to take into account the out-of-plane thickness. This is because the selected points correspond to edges in a 3D geometry.

You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

• Theory for Heat Transfer in Thin Structures
• Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Shells>Heat Source
Heat Transfer in Films>Heat Source
Heat Transfer in Fractures>Heat Source

Ribbon
Physics tab with **Heat Transfer in Shells** selected:

Edges>Heat Source

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.

MATERIAL TYPE
Select an option in the **Material type** list to specify if the inputs of the **Line Heat Source** section are defined in the material or spatial frame:

• The default option for the **Line Heat Source** node is **Solid**, which specifies that the heat source Q_l is defined in the material frame. Because the heat transfer variables and

equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See Conversion Between Material and Spatial Frames for details.

- The **Nonsolid** option defines Q_l in the spatial frame. No frame conversion is needed.

- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

📝 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 Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

LINE HEAT SOURCE

Select an option in the **Heat source** list:

- If **General source** (default) 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. This section is not available when the node is added under the **Thin Rod** feature.

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.

🔍 See Built-In Operators in the *COMSOL Multiphysics Reference Manual* for additional information about the diskavg operator.

> You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

> - Handling Frames in Heat Transfer
> - About the Heat Transfer Interfaces

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>*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 in Solids>*support***>Thin Rod>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**, or **Bioheat Transfer** selected:

*support***>Line Heat Source**
*support***>Thin Rod>Line Heat Source**

with *support* as **Egdes** in 3D and **Points** in 2D.

Lumped System Connector (Heat Transfer in Shells Interface)

Use this node to connect the distributed finite element model to a thermal network.

The average temperature on the selected edge can be prescribed at a node of the thermal network where an External Terminal feature of The Lumped Thermal System Interface is applied.

In return, the heat rate defined at the node of the thermal network is applied on the selected edge.

Select the edges on which to apply the connector. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, slmat1), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Lumped System Connector** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is **From physics interface** in the **Lumped System Connector** node. You can change to **User defined** and override the interface setting with a specific value or expression for L_{th}.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

@

- For a general description of layer and interface selections, see Layer and Interface Selection Tools.

- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

The desired selection for the node may correspond to boundaries with different layered materials.

🔍   See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

TERMINAL INPUTS

Select an **External Terminal** feature in the **Source** list to define which node of the thermal network provides the heat rate P_{ext} to be prescribed on the edge.

When the **Isothermal terminal** check box is not selected (default), the average temperature on the edge is constrained to the temperature at the terminal node of the thermal network. Select this check box to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Shells>Lumped System Connector

Ribbon
Physics tab with **Heat Transfer in Shells** selected:

Egdes>Lumped System Connector

Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)

Use this node to specify the side-by-side continuity conditions on the temperature field at an edge between two layered materials. By default, the temperature is discontinuous at the interface between two layered materials.

LAYER SELECTION

Select **Source** and **Destination** layered materials from the drop-down lists. This makes the node applicable on the edges adjacent to both selected layered materials. Click the **Layer Cross Section Preview** button to visualize the settings.

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

Q

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

Q

The definition of coinciding layer meshes at the continuity interface results in a more robust continuity condition. See the **Mesh elements** settings in the **Layer Definition** sections of the concerned **Single Layer Material** and **Layered Material**.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids>Shell Continuity
Heat Transfer in Shells>Continuity
Heat Transfer in Films>Continuity
Heat Transfer in Fractures>Continuity

Ribbon

Physics tab with **Heat Transfer in Solids** selected in the model tree:

Egdes>Shell Continuity

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Egdes>Continuity

Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

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

$$-\mathbf{n} \cdot \mathbf{q} \,=\, \varepsilon\sigma(T_{\text{amb}}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

EDGE SELECTION

Select the edges on which to add surface-to-ambient radiation. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, slmat1), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

SHELL PROPERTIES

Different settings are available, depending on the settings in the Shell Properties section of the parent interface or node:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Surface-to-Ambient Radiation** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Thickness** is taken **From physics interface** (or **From parent feature**) in the **Surface-to-ambient radiation** node. You can change to **User defined** and override the parent node setting with a specific value or expression for L_{th}.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

> The desired selection for the node may correspond to boundaries with different layered materials.
>
> See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

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 an **Ambient Properties** node under **Definitions**.

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 (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)
> - Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
> - Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
> - 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
Heat Transfer in Shells>Surface-to-Ambient Radiation

Ribbon
Physics tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:

Attributes>Surface-to-Ambient Radiation

Physics tab with **Heat Transfer in Shells** selected:

Egdes>Surface-to-Ambient Radiation

Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

This node provides a no flux condition for symmetry edges, similar to a Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface) condition.

It can be applied to exterior edges (3D) or points (2D and 2D axisymmetric) adjacent to the boundaries selected in the parent interface.

EDGE SELECTION

Select the exterior edges on which to apply the symmetry condition. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, slmat1), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

	The desired selection for the node may correspond to boundaries with different layered materials.
🔍	See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

SHELL PROPERTIES

This section is available only when the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface. You can limit the contribution to some layered materials (but not individually selected layers) by clearing the **Use all layers** check box.

LOCATION IN USER INTERFACE

Context Menus
Heat Transfer in Shells>Symmetry
Heat Transfer in Films>Symmetry
Heat Transfer in Fractures>Symmetry

Ribbon
Physics tab with **Heat Transfer in Shells** selected in the model tree:

Egdes>Symmetry

Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

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.

EDGE SELECTION

Select the edges on which to specify the temperature. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, `slmat1`), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

The desired selection for the node may correspond to boundaries with different layered materials.

See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

SHELL PROPERTIES

This section is available only when the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface or node. You can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

TEMPERATURE

For **User defined**, enter a value or expression for the **Temperature** T_0. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

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 More Options** button (👁) and select **Advanced Physics Options**.

📄	You can specify a harmonic variation of the temperature value in a frequency domain analysis of perturbation type, by adding a **Harmonic Perturbation** subnode. See Theory for Harmonic Heat Transfer for more details.

🔍	• Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) • Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface) • Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface) • 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 Solids>Thin Layer>Temperature

Heat Transfer in Solids>Thin Film>Temperature

Heat Transfer in Porous Media>Fracture>Temperature

Heat Transfer in Shells>Temperature

Heat Transfer in Films>Temperature

Heat Transfer in Fractures>Temperature

Ribbon

Physics tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:

Attributes>Temperature

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Egdes>Temperature

Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

This node is the default edge condition on shells. This condition means that there is no heat flux across the edge:

$$-\mathbf{n} \cdot \mathbf{q} = 0$$

and hence specifies where the shell is insulated. In most cases, the heat flux **q** is defined from the Fourier's law, and this condition sets the temperature gradient to zero across the edge. As the default edge condition, this feature is applicable only on external edges; but when added manually, it can be applied on interior edges as well.

At internal edges, and as a subfeature of **Thin Layer**, **Thin Film** or **Fracture** features, the thermal insulation condition can always be applied, but is only taken into account when the **Layer type** is **General** in the parent feature.

At internal edges corresponding to the junction of shells of different layered materials, if the **Layer type** is **General** in the **Thin Layer**, **Thin Film** or **Fracture** parent feature (Heat Transfer interface), or in the **Solid**, **Fluid** or **Porous Medium** feature (Heat Transfer in Shells interface), thermal insulation is implicit, and the temperature field is discontinuous across the edge, even if no **Thermal Insulation** subnode is active. This default behavior can be modified by using the Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface) feature on the edge.

EDGE SELECTION

Select the edges on which to apply a no flux condition. The **Restrict to layered edges** check box makes the node applicable only if a layered material is defined on the edge. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the edge index (for example, slmat1), otherwise the edge is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Restrict to layered boundaries** check box is not editable.

In addition, in 2D, edges are modeled as points and this section is named **Point Selection**, containing a **Restrict to layered points** check box.

SHELL PROPERTIES

This section is available only when the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface or node. You can limit the contribution to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

⚠	When the **Layer type** is **Thermally thin approximation**, and the shell is adjacent to domains where the temperature field is continuous, applying a thermal insulation on such a thermally thin edge has no effect.

🔍	The desired selection for the node may correspond to boundaries with different layered materials.
	See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual*.

🔍	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 Solids>Thin Layer>Thermal Insulation
Heat Transfer in Shells>Thermal Insulation

Ribbon
Physics tab with **Thin Layer** selected in the model tree:

Attributes>Thermal Insulation

Physics tab with **Heat Transfer in Shells** selected:

Egdes>Thermal Insulation

Thin Rod

Use this node to define the thermal and radius properties of conductive rods located on edges in a 3D component.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable minput.Tempref, which is set to 293.15 K by default. To edit it, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.

⚠️   This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

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 **Full** 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**. See Material Density in Features Defined in the Material Frame if a temperature-dependent density should be set. For **User defined** enter other values or expressions.

> These additional subnodes are available for the **Thin Rod** node:
>
> - Line Heat Source — 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 in Solids and Fluids>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

Ribbon
Physics tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, or **Bioheat Transfer** selected:

Edges>Thin Rod

Point Features

The Heat Transfer interfaces have the following point nodes and subnodes available:

- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis

- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Rod)

> 🔍 For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

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

The available options in the **Flux type** list are: **General inward heat flux** (default), **Convective heat flux**, and **Nucleate boiling heat flux**.

- 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 **Convective heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{ext} - T)$. Enter the **Heat transfer coefficient** h and the **External temperature** T_{ext}. The value depends on the geometry and the ambient flow conditions.

- If **Nucleate boiling heat flux** is selected, it adds q_0 estimated with the Rohsenow's correlation. See Nucleate Boiling Heat Flux for details about the settings.

> 🔍 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.

🔍	See Built-In Operators in the *COMSOL Multiphysics Reference Manual* for additional information about the `ballavg` and `diskavg` operators.

You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

- 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

Ribbon
Physics tab with **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, or **Bioheat Transfer** 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.

	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose **Harmonic Perturbation**. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.

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

$$\lim_{\partial S \to 0} \int_{\partial S} Q ds \ = \ d_{\mathrm{s}} \varepsilon \sigma (T_{\mathrm{amb}}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

MODEL INPUT
This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. 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** (![icon]( )). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (![icon]( )), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

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 an **Ambient Properties** node under **Definitions**.

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.

![search icon]( )	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 an **Ambient Properties** node under **Definitions**. 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 More Options** button (👁) and select **Advanced Physics Options**.

🔍	Thin Rod

📝	You can specify a harmonic variation of the temperature value in a frequency domain analysis of perturbation type, by adding a **Harmonic Perturbation** subnode. See Theory for Harmonic Heat Transfer for more details.

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

The Lumped Thermal System interface has the following global nodes available:

- Conductive Thermal Resistor
- Convective Thermal Resistor
- External Terminal
- Heat Pipe
- Heat Rate
- Heat Rate Source
- Nucleate Boiling Heat Rate
- Radiative Heat Rate

- Radiative Thermal Resistor
- Subsystem Definition
- Subsystem Instance
- Temperature
- Thermal Capacitor
- Thermal Mass
- Thermoelectric Module

🔍	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

Conductive Thermal Resistor

This feature models a two-port passive component of the thermal system. It connects two nodes, by creating a difference in the temperatures of its two connecting ports. It models heat loss by conduction in a domain.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the following relation between the heat rate P and the temperature difference ΔT:

$$P = -\frac{\Delta T}{R}$$

where R (SI unit: K/W) is the thermal resistance, which may be defined depending on thermal and geometric properties, and may take into account convection and radiation in optically thick participating medium.

See Theory for the Conductive Thermal Resistor Component for more details on the underlying theory.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

By default the **Temperature** is **User defined** and the average of the two port temperatures, `Tave = 0.5*(Tp1+Tp2)`, is set.

IDENTIFIER

Enter a **Component name** for the thermal resistor. The prefix is R.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the thermal resistor.

COMPONENT PARAMETERS

The thermal resistance used to express the heat rate P in function of the temperature difference ΔT should be defined in this section.

- When **Specify** is **Thermal resistance**, enter directly a value or expression for R.
- When **Specify** is **Thermal and geometric properties**, further settings display underneath to define the thermal resistance depending on the thermal conductivity and the geometric configuration.

Select any material from the **Material** list to define the **Thermal conductivity** k **From material**. For **User defined** enter a value or expression.

Select a **Configuration** among the following options, and set the needed geometric properties:

- **Plane shell** (default): set values or expressions for the surface **Area** A and the **Thickness** L of the plane. The thermal resistance is then defined as

$$R = \frac{L}{kA}$$

- **Cylindrical shell**: set values or expressions for the **Inner radius** r_i, the **Outer radius** r_o, and the **Height** H of the cylinder. The thermal resistance is then defined as

$$R = \frac{1}{2\pi k H} \ln\left(\frac{r_o}{r_i}\right)$$

- **Spherical shell**: set values or expressions for the **Inner radius** r_i and the **Outer radius** r_o of the sphere. The thermal resistance is then defined as

$$R = \frac{1}{4\pi k}\left(\frac{1}{r_i} - \frac{1}{r_o}\right)$$

When the **Configuration** is **Plane shell**, select the **Convectively enhanced conductivity** check box to take into account convective heat flux by enhancing the thermal conductivity according to the Nusselt number. Further settings (see below) are then required in the **Convectively Enhanced Conductivity** section that appears underneath.

For all **Configuration** options, select the **Optically thick participating medium** check box to take into account radiation in a medium with high optical thickness. Further settings (see below) are then required in the **Optically Thick Participating Medium** section that appears underneath.

CONVECTIVELY ENHANCED CONDUCTIVITY

This section is available when the **Convectively enhanced conductivity** check box is selected in the **Component Parameters** section. Convection is accounted for by multiplying the thermal conductivity by the Nusselt number.

The following options are available in the **Nusselt number correlation** list:

- **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. Unfold the **Sketch** section for more details on the required parameters.

- **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. Unfold the **Sketch** section for more details on the required parameters. By default, the **Thickness value** set in the **Component Parameters** section for L is used for the **Plate distance**.

- **User defined**, for which a value for Nu should be specified directly.

For the two first options, select **Automatic** (default) or **User defined** to define the **Temperature difference** ΔT. When **Automatic** is selected the temperature difference across the component is used.

Select a **Fluid type** between **Gas/Liquid** and **Ideal gas**, and depending on the selected option, set values or expressions for the material properties needed to calculate the Nusselt number. When the properties are taken **From material**, the material selected in the **Component Parameters** section is used. If the material properties are temperature-dependent, they are evaluated at the average temperature in the component, Tave = 0.5*(Tp1+Tp2).

🔍   Equivalent Thermal Conductivity Correlations

OPTICALLY THICK PARTICIPATING MEDIUM

This section is available when the **Optically thick participating medium** check box is selected in the **Component Parameters** section. It defines the properties of the participating medium to model the heating due to the propagation of the rays by modifying the thermal conductivity with

$$k_{\mathrm{R}} = \frac{16 n_{\mathrm{r}}^2 \sigma_s T^3}{3 \beta_{\mathrm{R}}}$$

where n_{r} is the refractive index (dimensionless), σ_{s} is the Stefan-Boltzmann constant (SI unit: $W/(m^2 \cdot K^4)$), and β_{R} is the extinction coefficient. The settings are the same as for the Optically Thick Participating Medium feature.

When the properties are taken **From material**, the material selected in the **Component Parameters** section is used. If the material properties are temperature-dependent, they are evaluated at the average temperature in the component, Tave = 0.5*(Tp1+Tp2).

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\mathrm{init}}$, and the **Initial temperature at node 2**, $T_{2,\mathrm{init}}$, to be used at initialization, in particular to evaluate the material properties of the component.

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature at node 1**

- **Temperature at node 2**

	Lumped Composite Thermal Barrier: Application Library path
▥	**Heat_Transfer_Module/Tutorials,_Thin_Structure/**
	lumped_composite_thermal_barrier

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Conductive Thermal Resistor

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Conductive Thermal Resistor

Convective Thermal Resistor

This feature models a two-port passive component of the thermal system. It connects two nodes, by creating a difference in the temperatures of its two connecting ports. It models heat loss by convection in a fluid of temperature T_{p2} over a solid wall of temperature T_{p1}.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the following relation between the heat rate P and the temperature difference ΔT:

$$P = -\frac{\Delta T}{R_{\text{conv}}}$$

where R_{conv} (SI unit: K/W) is the thermal resistance, which is defined from the surface **Area** A and the **Heat transfer coefficient** h as

$$R_{\text{conv}} = \frac{1}{hA}$$

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

By default the **Temperature** is **User defined** and set to the port p1 temperature, Tvar = Tp1.

IDENTIFIER

Enter a **Component name** for the convective thermal resistor. The prefix is CR.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the convective thermal resistor. Note that the ports p1 and p2 correspond respectively to the solid wall and fluid regarding the evaluation of the temperature.

COMPONENT PARAMETERS

Set the **Area** A of the surface of the convective heat rate and define the **Heat transfer coefficient** h used to express the convective thermal resistance. 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 rate 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** (default), **Transformer oil**, **Water**, **Moist air**, or **From material**.

When **From material** is selected, choose a material available from the **Materials** list.

Depending of the selected option, different parameters are needed. You can refer to the **Sketch** section to get an illustration of the configuration.

See the Fluid feature for more details on the settings related to each correlation.

The external conditions should be set.

Set the **Absolute pressure**, p_A. For **User Defined**, enter a value or expression. Else, select an **Ambient absolute pressure** defined in an **Ambient Properties** node under **Definitions**. The pressure is used to evaluate the **Fluid** material properties and this setting is not available for the **Transformer oil** and **Water** options.

Finally, when the **Fluid** is **Moist air**, also set the **External relative humidity**, $\phi_{w, ext}$, and the **Surface relative humidity**, $\phi_{w, s}$, used to evaluate the material properties.

For the external temperature, the temperature T_{p2} at the connecting port p2 is used.

	• This feature is the two-port component equivalent of the Heat Rate node with **Convective heat rate** option.
$\mathbb{Q}$	• The detailed definition of the predefined heat transfer coefficients is given in Convective Heat Transfer Correlations.
	• For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 21.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,init}$, and the **Initial temperature at node 2**, $T_{2,init}$, to be used at initialization, in particular to evaluate the heat transfer coefficient.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

• **Heat rate**

• **Temperature at node 1**

• **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Convective Thermal Resistor

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Convective Thermal Resistor

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 factors calculation. Only the external view factors (F_{ext}) are recomputed when the radiation source moves, not the complete

view factors. When specular radiation is involved (with **Opaque Surface** or **Semitransparent Surface**), the cost of computation of these external view factors can be important. 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**, **Diffuse Mirror, Opaque Surface** or **Semitransparent Surface** boundary condition is active. The source contribution, $G_{\mathrm{extDir},\, i}$, is equal to the product of the view factor of the source by the source radiosity. For radiation sources located on a point, $G_{\mathrm{extDir},\, i} = F_{\mathrm{ext},\, i}\, P_{\mathrm{s},\, i}$. For directional radiative source, $G_{\mathrm{extDir},\, i} = F_{\mathrm{ext},\, i}\, q_{0,\, \mathrm{s}}$.

Only direct irradiation from the source is accounted for. Diffuse irradiation inclusion is controlled through the **Include diffuse radiation** check box in **Ambient** section of Diffuse Surface (Surface-to-Surface Radiation Interface) and Diffuse Mirror (Surface-to-Surface Radiation Interface) features.

📄	The external radiation sources are ignored on the boundaries when neither **Diffuse Surface**, **Diffuse Mirror, Opaque Surface** nor **Semitransparent Surface** is active.

EXTERNAL RADIATION SOURCE

Select a **Source position**: **Point coordinate** (default) or **Infinite distance**. In 3D, **Solar position** is also available.

Point Coordinate
For **Point coordinate** define the **Source location $\mathbf{x}_{\mathrm{s}}$**. The source radiates uniformly in all directions.

❗	$\mathbf{x}_{\mathrm{s}}$ should not belong to any surface where a **Diffuse Surface**, **Diffuse Mirror, Opaque Surface** or **Semitransparent Surface** boundary condition is active.

Infinite Distance
For **Infinite distance** define the **Incident radiation direction $\mathbf{i}_{\mathrm{s}}$**.

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 direct striking of the Sun rays.

North, west, and the up directions correspond to the *x*, *y*, and *z* directions, respectively. Azimuth angle is measured from true north, hence *x* direction corresponds to true north as well.

Depending on the presence of an **Ambient Properties** node under **Definitions** and on the type of **Ambient data** selected in this node, further parameters should be set for the definition of the location on earth.

If the type of **Ambient data** is **Meteorological data (ASHRAE 2013)** or **Meteorological data (ASHRAE 2017)** in the **Ambient Properties** node under **Definitions**, it is available in the **Ambient data** list of the **External Radiation Source** node. When selected from this list, the location is set to the **Weather station** selected in the **Ambient Properties** node under **Definitions**. Click to select the **Include daylight saving time (Time zone + 1)** check box to add one hour to the default setting for the station selected.

Else, when **Ambient data** is **User defined** in the **Ambient Properties** node under **Definitions**, **None** is the only option in the **Ambient data** list of the **External Radiation Source** node, and the following parameters should be set.

Select an option from the **Location defined by** list: **Coordinates** (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 (**City** or **Coordinates**), 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 (**City** or **Coordinates**), 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 start 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. Clear this check box to manually set the time update.

For either selection of **Ambient data** type in an **Ambient Properties** node under **Definitions**, define the **Solar irradiance** field I_s as the incident radiative intensity coming directly 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 a **Clear sky noon beam normal irradiance** defined in an **Ambient Properties** node under **Definitions**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the solar irradiance is divided among all spectral bands B_i as $q_{s,i} = q_{0,s}FEP_i(T_{sun})$ where $FEP_i(T_{sun})$ is the fractional blackbody emissive power over B_i interval at $T_{sun} = 5780$ K.

RADIATIVE INTENSITY

This section is available when **Source position** is set either to **Point coordinate** or **Infinite distance**.

Point Coordinate

If **Wavelength dependence of radiative properties** is **Constant**, enter a value or expression to define the **Source heat rate** P_s.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, set the **Radiative intensity** to **Blackbody**, **User defined for each band**, or **User defined**.

When **Radiative intensity** is set to **User defined**, enter a value or expression for the **Source heat rate distribution** $P_{s,\lambda}$. The wavelength may be accessed through the `rad.lambda` variable. This distribution is integrated on each spectral band to obtain the source heat rate $P_{s,i}$ for each spectral band.

When **Radiative intensity** is set to **Blackbody**, enter a value for the **Source temperature** T_s and the **Source heat rate** P_s, to define the source power on the spectral band B_i as $P_{s,i} = FEP_i(T_s)P_s$ where $FEP_i(T_s)$ is the fractional blackbody emissive power over B_i interval at T_s.

When **Radiative intensity** is set to **User defined for each band**, enter a value for the **Source heat rate** $P_{s,i}$ for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength.

Infinite Distance

If **Wavelength dependence of radiative properties** is **Constant**, enter a value or expression to define the **Source heat flux** $q_{0,s}$. Alternatively, select a **Clear sky noon beam normal irradiance** defined in an **Ambient Properties** node under **Definitions**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, set the **Radiative intensity** to **Blackbody**, **User defined for each band**, or **User defined**.

When **Radiative intensity** is set to **User defined**, enter a value or expression for the **Source heat flux distribution** $q_{0,\mathrm{s},\lambda}$. The wavelength may be accessed through the rad.lambda variable. This distribution is integrated on each spectral band to obtain the source heat flux $q_{0,\mathrm{s},i}$ for each spectral band.

When **Radiative intensity** is set to **Blackbody**, enter a value for the **Source temperature** T_s and the **Source heat flux** $q_{0,\mathrm{s}}$, to define the source heat flux on the spectral band B_i as $q_{\mathrm{s},i} = FEP_i(T_\mathrm{s}) q_{0,\mathrm{s}}$ where $FEP_i(T_\mathrm{s})$ is the fractional blackbody emissive power over B_i interval at T_s.

When **Radiative intensity** is set to **User defined for each band**, enter a value for the **Source heat flux** $q_{0,\mathrm{s},i}$ for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength.

📑	The **Wavelength dependence of radiative properties** is defined in the physics interface settings, in the Radiation Settings section. When only one spectral band is defined, the i 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 (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 calculated for more than 60s.

🔍	The 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**

Symmetry for Surface-to-Surface Radiation

Use this node to calculate view factors on only a part of a symmetric geometry to improve efficiency, by defining either one symmetry plane in 2D, 2D axisymmetric, and 3D components, multiple perpendicular symmetry planes in 2D 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 6-6 summarizes the available options for each dimension.

TABLE 6-6: OPTIONS FOR SYMMETRY DEFINITION, EXAMPLE WITH 5 SECTORS
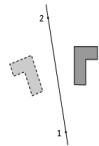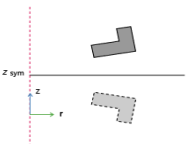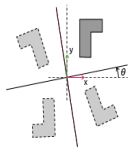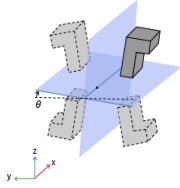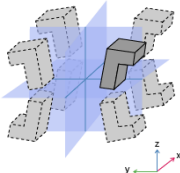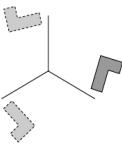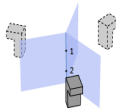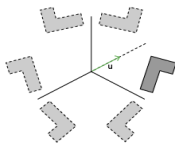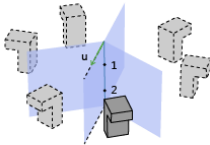
SYMMETRY TYPE	2D	2D AXISYMMETRIC	3D
One Plane			
Two Planes		Not applicable	

TABLE 6-6: OPTIONS FOR SYMMETRY DEFINITION, EXAMPLE WITH 5 SECTORS

SYMMETRY TYPE	2D	2D AXISYMMETRIC	3D
Three Planes	Not applicable	Not applicable	
Sectors		Not applicable	
Sectors and Reflection		Not applicable	

SYMMETRY FOR SURFACE-TO-SURFACE RADIATION

Depending on component's dimension, select the **Type of symmetry** to be defined:

- In 2D components, select between **Plane of symmetry** (default), **Two perpendicular planes of symmetry**, or **Sectors of symmetry**.

- In 2D axisymmetric components, only the **Plane of symmetry** option is available.

- In 3D components, select between **Plane of symmetry** (default), **Two perpendicular planes of symmetry**, **Three perpendicular planes of symmetry**, 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

The coordinates of the points defining the plane should be set:

- In 2D components, the symmetry plane is defined by two points. If the **Selection method** is **Coordinates**, set the **x** and **y** coordinates of the **First point on plane of reflection** and of the **Second point on plane of reflection**. Else, if the **Selection method** is **Point Selection**, you can directly select the points from the **Graphics** window. In the **Selection of First Point** and **Selection of Second Point** sections, first use the **Active** button to toggle between turning ON ⬛ and OFF ⬜ selections, that is, making the selections active for that selection list. Then select the points from the **Graphics** window.

- In 2D axisymmetric components, the symmetry plane is parallel to the $z=0$ plane. If the **Selection method** is **Coordinates**, set the **z coordinate of plane of reflection**, z_{sym}. Else, if the **Selection method** is **Point Selection**, you can directly select the point from the **Graphics** window. In the **Selection of the Point Defining the Plane of Reflection** section, first use the **Active** button to toggle between turning ON ⬛ and OFF ⬜ selections, that is, making the selection active for that selection list. Then select the point from the **Graphics** window.

- In 3D components, the symmetry plane is defined by three points. If the **Selection method** is **Coordinates**, set the **x**, **y**, and **z** coordinates of the **First point on plane of reflection**, the **Second point on plane of reflection**, and the **Third point on plane of reflection**. Else, if the **Selection method** is **Point Selection**, you can directly select the points from the **Graphics** window. In the **Selection of First Point**, **Selection of Second Point**, and **Selection of Third Point** sections, first use the **Active** button to toggle between turning ON ⬛ and OFF ⬜ selections, that is, making the selections active for that selection list. Then select the points from the **Graphics** window.

The **Show the plane of symmetry** check box is selected by default. Clear it to hide the plane in the **Graphics** window.

Two Perpendicular Planes of Symmetry

- In 2D components, the planes intersection line is along the out-of-plane direction. The **Point at the intersection of symmetry planes** and the **Rotation angle** from Cartesian axis, θ, should be defined.

- In 3D components, the **Planes intersection** line can be set **Along the x-axis**, **Along the y-axis**, or **Along the z-axis**. A **Point at the intersection of symmetry planes** and the **Rotation angle** from Cartesian axis, θ, should be defined. The position of the point along the intersection line can be arbitrary as it does not change the planes

definition. The direction of the angle θ can be checked on the figure displayed below the settings, for each case of alignment of the intersection line.

The **Show the planes of symmetry** check box is selected by default. Clear it to hide the planes in the **Graphics** window.

Three Perpendicular Planes of Symmetry
The planes are parallel to the Cartesian axis. Only the **Point at the intersection of symmetry planes** should be defined.

The **Show the planes of symmetry** check box is selected by default. Clear it to hide the planes in the **Graphics** window.

Sectors of Symmetry
The coordinates of the points defining the symmetry axis should 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**.

- 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** and the **Second point defining sector symmetry axis**.

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**, **u**. This option can be used when each sector has itself a plane of symmetry.

Q
- The Surface-to-Surface Radiation Interface
- Theory for Surface-to-Surface Radiation

LOCATION IN USER INTERFACE

Context Menus
Surface-to-Surface Radiation>Symmetry for Surface-to-Surface Radiation

Ribbon
Physics tab with **Surface-to-Surface Radiation** selected:

Global>Symmetry for Surface-to-Surface Radiation

External Terminal

Use this feature to connect a node (one-port component) of the thermal system to a distributed finite element model. It prescribes the temperature T_{ext} provided by a Lumped System Connector boundary feature of a Heat Transfer interface, a Lumped System Connector, Interface (Heat Transfer in Shells Interface) boundary interface feature of a Heat Transfer in Shells interface, or a Lumped System Connector (Heat Transfer in Shells Interface) edge interface feature of a Heat Transfer in Shells interface.

The **Lumped System Connector** feature uses the heat rate defined by the **External Terminal** feature on the selected node to set a heat flux on the corresponding boundary, interface, or edge in the distributed finite element model.

NODE CONNECTIONS

Set the **Node name** for the external terminal.

EXTERNAL CONNECTION

This section displays an information message about the connection status of the node:

- If the **External Terminal** feature is selected in the **Source** list of a **Thermal Connector** feature, the message **Current status: connected** is displayed.

- Else, the message **Current status: not connected** is displayed. The default value 293.15[K] is used for T_{ext}.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature**

▥	*Lumped Composite Thermal Barrier*: Application Library path **Heat_Transfer_Module/Tutorials,_Thin_Structure/ lumped_composite_thermal_barrier**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Nodes>External Terminal

Heat Pipe

This feature models a two-port passive component of the thermal system used to take into account heat loss in a heat pipe. It connects two nodes, by creating a difference in the temperatures of its two connecting ports, corresponding to the evaporator and condenser sides of the heat pipe.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the following relation between the heat rate P and the temperature difference ΔT:

$$P = -\frac{\Delta T}{R}$$

where R (SI unit: K/W) is the total thermal resistance, accounting for the relevant heat transfer processes present in the heat pipe. It may be defined either directly or as the sum of the resistances related to conduction through the solid wall and porous wick of the heat pipe, depending on the available data for the thermal and geometric properties.

Note that $\Delta T = T_{condenser} - T_{evaporator}$ and is meant to be negative in the classical operating mode of the heat pipe. The heat rate P is set to 0 when this condition does not hold.

See Theory for the Heat Pipe Component for more details on the underlying theory.

MODEL INPUT
This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

By default the **Temperature** is **User defined** and the average of the two port temperatures, `Tave = 0.5*(Tp1+Tp2)`, is set.

IDENTIFIER
Enter a **Component name** for the heat pipe. The prefix is HP.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the heat pipe. Note that the ports p1 and p2 correspond respectively to the evaporator and condenser sides of the heat pipe.

COMPONENT PARAMETERS

Select an option from the **Model** list for the expression of the total thermal resistance R:

- when **Total resistance** is selected, set a value or expression for the **Total thermal resistance**, R.

- when **Multiple inputs** (default) is selected, the total thermal resistance is the sum of the thermal resistances due to conduction in the wall and the wick of the heat pipe, on the evaporator and condenser sides, and further settings are required.

When **Input quantity** is set to **Thermal resistances**, enter values or expressions for the **Thermal resistance, evaporator wall**, $R_{\mathrm{wall,e}}$, the **Thermal resistance, evaporator wick**, $R_{\mathrm{wick,e}}$, the **Thermal resistance, condenser wick**, $R_{\mathrm{wick,c}}$, and the **Thermal resistance, condenser wall**, $R_{\mathrm{wall,c}}$.

In the network representation of heat transfer in the heat pipe, these thermal resistances are connected in a serial way, and the total thermal resistance R is thus defined as:

$$R \;=\; R_{\mathrm{wall,\,e}} + R_{\mathrm{wick,\,e}} + R_{\mathrm{wick,\,c}} + R_{\mathrm{wall,\,c}}$$

The thermal resistances due to transport of vapor in the channel and transport of liquid in the porous wick are neglected in this network representation, see Theory for the Heat Pipe Component for details.

Alternatively, set **Input quantity** to **Configuration and thermal properties** to calculate the thermal resistances from the geometric dimensions and thermal properties of the wick and wall of the heat pipe. This option is available for a flat or cylindrical heat pipe. Further settings display underneath to define the geometric configuration. You can refer to the **Sketch** section to get an illustration of the configuration.

Cylindrical

Set values or expressions for the geometric dimensions used to calculate the wick and wall thermal resistances on the evaporator and condenser sides: **Outer radius**, r_{o}, **Wall thickness**, d_{wall}, **Wick thickness**, d_{wick}, **Evaporator length**, L_{e}, and **Condenser length**, L_{c}.

Flat

Set values or expressions for the geometric dimensions used to calculate the wick and wall thermal resistances on the evaporator and condenser sides: **Depth**, D, **Height**, H, **Wall thickness**, d_{wall}, **Wick thickness**, d_{wick}, **Evaporator length**, L_{e}, and **Condenser length**, L_{c}.

HEAT CONDUCTION, WALL

The thermal conductivity k_{s} of the wall should be set in this section when **Input quantity** is set to **Configuration and thermal properties** in the **Component Parameters** section.

Select any material from the **Material** list to define the **Thermal conductivity** k **From material**. Alternatively, choose among **Copper** and **Aluminum** for a predefined material. For **User defined** enter a value or expression.

HEAT CONDUCTION, WICK

The thermal conductivity k_{eff} of the wick should be set in this section when **Input quantity** is set to **Configuration and thermal properties** in the **Component Parameters** section.

The wick is a porous medium. When **Input quantity** is set to **Structure model**, its effective thermal conductivity is expressed as a function of its porosity and the thermal conductivities of the immobile solid and the liquid parts.

Select any material from the **Material** list to define the **Thermal conductivity, liquid** k_{l} **From material**. Alternatively, choose among **Water** and **Mercury** for a predefined material. For **User defined** enter a value or expression.

Select any material from the **Material** list to define the **Thermal conductivity, immobile** k_{w} **From material**. Alternatively, choose among **Copper** and **Aluminum** for a predefined material. For **User defined** enter a value or expression.

Then, set a value or expression for the **Porosity**, ε, and select an option from the **Structure** list to calculate the effective conductivity. See Theory for the Heat Pipe Component for details.

Alternatively, set **Input quantity** to **User defined** to set directly the **Effective thermal conductivity of the wick**, k_{eff}.

Operating Maximum Power

Finally, select the **Specify operating maximum power** check box to curb the heat pipe when the operating maximum power P_{max} is known. Set a value or expression for P_{max}.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,init}$, and the **Initial temperature at node 2**, $T_{2,init}$, to be used at initialization, in particular to evaluate the material properties of the heat pipe.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature at node 1**
- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Heat Pipe

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Heat Pipe

Heat Rate

This feature models a node (one-port component) of the thermal system, at which the heat rate is prescribed.

NODE CONNECTIONS

Set the **Node name** for the heat rate node.

NODE PARAMETERS

Select an option from the **Specify** list: **Heat rate** (default), **Convective heat rate**, **Nucleate boiling heat rate**, or **Joule heating**.

Heat Rate

Enter a value or expression for P_0 to prescribe it as the heat rate at the selected node of the thermal system.

Convective Heat Rate

The heat rate P is defined in function of the difference between the node temperature T and a specified external temperature, T_{ext}, by mean of a convective thermal resistance, R_{conv}:

$$P = \frac{T_{ext} - T}{R_{conv}}$$

Set the **Area** A of the surface of the convective heat flux and define the **Heat transfer coefficient** h used to express the convective thermal resistance as:

$$R_{conv} = \frac{1}{hA}$$

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 rate 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** (default), **Transformer oil**, **Water**, **Moist air**, or **From material**.

When **From material** is selected, choose a material available from the **Materials** list.

Depending of the selected option, different parameters are needed. You can refer to the **Sketch** section to get an illustration of the configuration.

See the Fluid feature for more details on the settings related to each correlation.

Finally the external conditions should be set.

Set the **Absolute pressure**, p_A. For **User Defined**, enter a value or expression. Else, select an **Ambient absolute pressure** defined in an **Ambient Properties** node under **Definitions**. The pressure is used to evaluate the **Fluid** material properties and this setting is not available for the **Transformer oil** and **Water** options.

In addition, enter an **External temperature**, T_{ext}. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

Finally, when the **Fluid** is **Moist air**, also set the **External relative humidity**, $\phi_{w,\,ext}$, and the **Surface relative humidity**, $\phi_{w,\,s}$, used to evaluate the material properties.

⚲	• The detailed definition of the predefined heat transfer coefficients is given in Convective Heat Transfer Correlations. • For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 21.

Nucleate Boiling Heat Rate
The heat rate P is defined as follows:

$$P = Aq_0$$

where A is surface area, and q_0 is computed with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See Nucleate Pool Boiling Correlation for details about the correlation. Set the **Area** A of the surface of the nucleate boiling heat flux, and select materials from the **Fluid** and **Surface** lists. When a predefined material (**Water**, **Benzene**, **n-Pentane**, or **Ethanol**) is used for the fluid, no additional parameters are required, because the liquid and vapor properties are predefined (at saturation temperature) and the Rohsenow's correlation parameters C_{sf} and s are available for different surface materials. Else, see Nucleate Boiling Heat Flux (Heat Flux feature) for details about the settings.

Joule Heating
The heat rate P is defined as follows to model resistive heating at the node:

$$P = R_e I^2$$

where R_e is the electrical resistance and I is the electric current.

Set values or expressions for the **Electric current**, I, and the **Electrical resistance**, R_e.

INITIAL VALUES

Set a user defined value or expression for the **Initial temperature**, T_{init}, to be used at initialization, in particular to evaluate the heat transfer coefficient.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Nodes>Heat Rate

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Nodes>Heat Rate

Heat Rate Source

This feature models a two-port source component of the thermal system. It connects two nodes, by creating a difference in the heat rates of its two connecting ports. It models a heat source or sink in a domain.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the heat rate P.

IDENTIFIER

Enter a **Component name** for the heat rate source. The prefix is P.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the heat rate source.

COMPONENT PARAMETERS

Set a value or expression for the heat rate P_{src}.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$.

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature at node 1**

- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Heat Rate Source

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Heat Rate Source

Nucleate Boiling Heat Rate

This feature models a two-port source component of the thermal system. It connects two nodes, by creating a difference in the heat rates of its two connecting ports. It models a heat sink due to nucleate boiling.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the heat rate P_{src}.

IDENTIFIER

Enter a **Component name** for the nucleate boiling heat rate. The prefix is NBP.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the nucleate boiling heat rate.

COMPONENT PARAMETERS

The heat rate P_{src} is defined as follows:

$$P_{\text{src}} = Aq_0$$

where A is surface area, and q_0 is computed with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See Nucleate Pool Boiling Correlation for details about the correlation. Set the **Area** A of the surface of the nucleate boiling heat flux, and select materials from the **Fluid** and

Surface lists. When a predefined material (**Water**, **Benzene**, **n-Pentane**, or **Ethanol**) is used for the fluid, no additional parameters are required, because the liquid and vapor properties are predefined (at saturation temperature) and the Rohsenow's correlation parameters C_{sf} and s are available for different surface materials. Else, see Nucleate Boiling Heat Flux (Heat Flux feature) for details about the settings.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature at node 1**

- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Nucleate Boiling Heat Rate

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Nucleate Boiling Heat Rate

Radiative Heat Rate

This feature models a node (one-port component) of the thermal system, at which a heat rate due to surface-to-ambient radiation is prescribed.

NODE CONNECTIONS

Set the **Node name** for the radiative heat rate node.

NODE PARAMETERS

The heat rate P due to radiation on a diffuse-gray surface from the ambient surrounding is:

$$P = \varepsilon\sigma A(T_{\text{amb}}^4 - T^4)$$

where σ is the Stefan-Boltzmann constant, ε is the emissivity of the surface, A is the area of the surface, T_{amb} is the ambient temperature, and T is the surface temperature.

In the thermal system, the corresponding heat rate applied on the selected node can be expressed in function of the difference between the ambient temperature, T_{amb} and the node temperature, T, by mean of the radiative thermal resistance, R_{rad}:

$$P = \frac{T_{\text{amb}} - T}{R_{\text{rad}}}$$

where

$$R_{\text{rad}} = \frac{1}{\varepsilon\sigma A(T_{\text{amb}}^2 + T^2)(T_{\text{amb}} + T)}$$

First, set the surface **Area** A. Then, select any material from the **Material** list to define the **Surface emissivity** ε **From material**. For **User defined** enter a value or expression.

Finally, set the **Ambient temperature**, T_{amb}. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

INITIAL VALUES

Set a user defined value or expression for the **Initial temperature**, T_{init}, to be used at initialization, in particular to evaluate the surface emissivity.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Nodes>Radiative Heat Rate

Ribbon
Physics tab with **Lumped Thermal System** selected:

Radiative Thermal Resistor

This feature models a two-port passive component of the thermal system. It connects two nodes, by creating a difference in the temperatures of its two connecting ports. It models heat loss by surface-to-surface radiation in a domain.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the following relation between the heat rate P and the temperature difference ΔT:

$$P = -\frac{\Delta T}{R}$$

where R (SI unit: K/W) is the radiative thermal resistance, which is defined to take into account radiative heat transfer between two opaque diffuse gray surfaces in an enclosure.

See Theory for the Radiative Thermal Resistor Component for more details on the underlying theory.

IDENTIFIER

Enter a **Component name** for the radiative thermal resistor. The prefix is RR.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the radiative thermal resistor.

COMPONENT PARAMETERS

Select a **Radiation type** for the surfaces. By default, **Blackbody radiation** is assumed, which sets $\varepsilon_1=\varepsilon_2=1$ in the expression of the radiative thermal resistance R. When the **Radiation type** is **Graybody radiation**, select any material from the **Material** lists to define the **Surface emissivity** ε_1 and **Surface emissivity** ε_2 **From material**. For **User defined** enter values or expressions.

Select a **Surface configuration** from the following options:

- **User defined** (default): specify the **View factor**, F_{12}, and the **Area** of surface $1, A_1$, used in the expression of the radiative thermal resistance.
- **Large (infinite) parallel planes**: specify the **Area** of surface $1, A_1$, used in the expression of the radiative thermal resistance. In this configuration, $F_{12}=1$.

- **Long (infinite) concentric cylinders**: specify the **Height** of the cylinders, H, the **Radius** of the **Inner cylinder**, r_1, and the **Radius** of the **Outer cylinder**, r_2, used in the expression of the radiative thermal resistance. In this configuration, $F_{12}=1$.

- **Concentric spheres**: specify the **Radius** of the **Inner sphere**, r_1 and the **Radius** of the **Outer sphere**, r_2, used in the expression of the radiative thermal resistance. In this configuration, $F_{12}=1$.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$, to be used at initialization, in particular to evaluate the surface emissivities.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature at node 1**

- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Radiative Thermal Resistor

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Radiative Thermal Resistor

Subsystem Definition

This feature is used to define subsystems, which can be inserted as components into the main system using a Subsystem Instance node. Create the subsystem by adding subnodes to the **Subsystem Definition** node.

NODE CONNECTIONS

Define the number of connecting nodes by adding more nodes or deleting existing nodes and set the **Node names** at which the subsystem connects to the main system or

to other subsystems. The order in which the **Node names** are defined is the order in which they are referenced by a **Subsystem Instance** node.

The components constituting the subsystem should be connected only to the subsystem's nodes and to themselves.

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Subsystems>Subsystem Definition

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Subsystems>Subsystem Definition

Subsystem Instance

This feature represents an instance of a subsystem defined by a Subsystem Definition feature.

IDENTIFIER

Enter a **Component name** for the subsystem instance. The prefix is X.

NODE CONNECTIONS

Select the **Name of subsystem link** from the list of defined subsystems in the model and set the **Node names** at which the subsystem instance connects to the main system or to another subsystem.

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Subsystems>Subsystem Instance

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Subsystems>Subsystem Instance

Temperature

This feature models a node (one-port component) of the thermal system, used to prescribe the temperature.

NODE CONNECTIONS

Set the **Node name** for the temperature node.

NODE PARAMETERS

Set a value or expression for the **Temperature**, T_0.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Nodes>Temperature

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Nodes>Temperature

Thermal Capacitor

This feature models a two-port passive component of the thermal system. It connects two nodes, by creating a difference in the temperatures of its two connecting ports. It models heat storage in a domain.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the following relation between the heat rate P and the temperature difference ΔT:

$$P = -C\frac{\partial \Delta T}{\partial t} \tag{6-20}$$

where C (SI unit: J/K) is the thermal capacitance, defined as

$$C = V\rho C_p = mC_p$$

with V the volume (SI unit: m^3), ρ the density (SI unit: kg/m^3), C_p the heat capacity at constant volume (SI unit: J/(kg·K)), and m the mass (SI unit: kg).

For a steady-state problem the temperature difference does not change with time and P=0.

See Theory for the Thermal Capacitor and Thermal Mass Components for more details on the underlying theory.

IDENTIFIER

Enter a **Component name** for the thermal capacitor. The prefix is C.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the thermal capacitor.

COMPONENT PARAMETERS

The thermal capacitance used in Equation 6-20 should be set in this section.

Depending on the option selected in the **Specify** list, different settings are required:

- With **Thermal capacitance** (default), set directly a value or expression for C.

- With **Density and heat capacity**, set values or expressions for the **Volume**, V, the **Density**, ρ, and the **Heat capacity at constant volume**, C_p. Select any material from the **Material** list to define the properties **From material**. For **User defined** enter values or expressions.

- With **Mass and heat capacity**, set values or expressions for the **Mass**, M, and the **Heat capacity at constant volume**, C_p. Select any material from the **Material** list to define C_p **From material**. For **User defined** enter a value or expression.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$, to be used at initialization, in particular to evaluate the heat storage term.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature at node 1**

- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Thermal Capacitor

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Thermal Capacitor

Thermal Mass

This feature models a node (one-port component) of the thermal system, at which a heat rate P is prescribed to take into account heat storage:

$$P = -C\frac{\partial T}{\partial t} \tag{6-21}$$

where C (SI unit: J/K) is the thermal capacitance, defined as

$$C = V\rho C_p = mC_p$$

with V the volume (SI unit: m^3), ρ the density (SI unit: kg/m^3), C_p the heat capacity at constant volume (SI unit: J/(kg·K)), and m the mass (SI unit: kg).

This component is similar to the Thermal Capacitor two-port component, at the difference that the thermal mass has a virtual fixed temperature node.

For a steady-state problem the temperature does not change with time and $P{=}0$.

See Theory for the Thermal Capacitor and Thermal Mass Components for more details on the underlying theory.

NODE CONNECTIONS
Set the **Node name** for the thermal mass node.

NODE PARAMETERS
The thermal capacitance used in Equation 6-21 should be set in this section.

Depending on the option selected in the **Specify** list, different settings are required:

- With **Thermal capacitance** (default), set directly a value or expression for C.

- With **Density and heat capacity**, set values or expressions for the **Volume**, V, the **Density**, ρ, and the **Heat capacity at constant volume**, C_p. Select any material from the **Material** list to define the properties **From material**. For **User defined** enter values or expressions.

- With **Mass and heat capacity**, set values or expressions for the **Mass**, M, and the **Heat capacity at constant volume**, C_p. Select any material from the **Material** list to define C_p **From material**. For **User defined** enter a value or expression.

INITIAL VALUES

Set a user defined value or expression for the **Initial temperature**, T_{init}, to be used at initialization, in particular to evaluate the heat storage term.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**

- **Temperature**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Nodes>Thermal Mass

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Nodes>Thermal Mass

Thermoelectric Module

This feature models a two-port source component of the thermal system, used to take into account Peltier effect and Joule heating in a thermoelectric module. It connects two nodes, by creating a difference in the heat rates of its two connecting ports.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the heat rate P as the sum of the heat rates P_{p1} and P_{p2}.

The heat rate applied at each port takes into account the sum of Peltier (cooling or heating) effect, half of the Joule heating, the resistive heating due to conduction in the thermocouples (p-type and n-type semiconductors), and heat storage for time-dependent modeling:

- p1 side:

$$P_{p1} = SIT_{p1} + \frac{R_e I^2}{2} - \frac{\Delta T}{R} - C_1 \frac{\partial T_{p1}}{\partial t} \qquad (6\text{-}22)$$

- p2 side:

$$P_{p2} = -SIT_{p2} + \frac{R_e I^2}{2} + \frac{\Delta T}{R} - C_2 \frac{\partial T_{p2}}{\partial t} \qquad (6\text{-}23)$$

where

- S (SI unit: V/K) is the Seebeck coefficient of the thermocouples
- I (SI unit: A) is the electric current operating in the module
- R_e (SI unit: Ω) is the electrical resistance of the thermocouples
- ΔT (SI unit: K) is the temperature difference between the two sides of the module
- R (SI unit: K/W) is the thermal resistance of the thermocouples
- C_1 and C_2 (SI unit: J/K) are the thermal capacitances on each side of the module

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

When summing Equation 6-22 and Equation 6-23 to obtain the total heat rate through the thermoelectric module, the terms for heat dissipation by conduction cancel each other. Note that the material properties are evaluated at the average temperature of the two ports temperatures.

See Theory for the Thermoelectric Module Component for details.

For a thermoelectric cooler, the ports p1 and p2 correspond respectively to the cold and hot sides of the module, and the cold side is the side where the heat should be removed.

In addition, the total heat rate through the thermoelectric module may be expressed from performance parameters:

- p1 side:

$$P_{\text{p1}} = -\left(Q_{max} - \Delta T \frac{Q_{max}}{\Delta T_{max}}\right) \qquad (6\text{-}24)$$

- p2 side:

$$P_{\text{p2}} = Q_{max} - \Delta T \frac{Q_{max}}{\Delta T_{max}} + R_e I^2 \qquad (6\text{-}25)$$

where

- Q_{max} (SI unit: W) is the maximum removed heat on the cold side
- ΔT (SI unit: K) is the temperature difference between the two sides of the module
- ΔT_{max} (SI unit: K) is the maximum temperature difference across the module
- R_{e} (SI unit: Ω) is the electrical resistance of the thermocouples
- I (SI unit: A) is the electric current operating in the module

This alternative formulation may be chosen if the material and geometric properties of the module are not available. See Performance Graphs for Thermoelectric Coolers for details.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

By default the **Temperature** is **User defined** and the average of the two port temperatures, `Tave = 0.5*(Tp1+Tp2)`, is set.

IDENTIFIER

Enter a **Component name** for the thermoelectric module. The prefix is TEM.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the thermoelectric module. Note that for a thermoelectric cooler, the ports p1 and p2 correspond respectively to the cold and hot sides of the module.

COMPONENT PARAMETERS

Depending on the available data for the specification of the thermoelectric module and on its operation mode (cooling or heating), different sets of user inputs may be entered:

- Set **Specify** to **Performance parameters, TEC linearized model** to use performance graphs (Equation 6-24 and Equation 6-25) to characterize the module. Then, set values or expressions for the operating **Electric current**, I, and set the other parameters in the Performance Parameters section.

- Set **Specify** to **Performance parameters, general model** to apply user defined heat rates on each side of the module. Set values or expressions for the **Heat rate, node 1** and the **Heat rate, node 2**. In this formulation, the nodes 1 and 2 may refer to any side of the module, and the heat rate expressions may depend on the temperature difference and on the current intensity, among other variables. The temperature difference can be accessed through the `lts.TEM1.DT` variable, when the component name is TEM1, in a physics interface with tag `lts`.

- Set **Specify** to **Thermal and geometric properties** to use Equation 6-22 and Equation 6-23 to characterize the module. Set values or expressions for the operating **Electric current**, I, and the **Number of thermocouples**, N (composed each of one p-type and one n-type semiconductors) in the thermoelectric module. Then the material and geometric properties should be set in the P-Type Semiconductor Parameters, N-Type Semiconductor Parameters, and Ceramic Plate Parameters section that display underneath.

PERFORMANCE PARAMETERS

This section is available when **Specify** is set to **Performance parameters, TEC linearized model** in the **Component Parameters** section.

Set the **Maximum heat rate, cold side**, $Q_{\max}$, the **Maximum temperature difference**, $\Delta T_{\max}$, and the **Electrical resistance**, R_{e}, used for the expression of the heat rates on each side, following Equation 6-24 and Equation 6-25.

P-TYPE SEMICONDUCTOR PARAMETERS

This section is available when **Specify** is set to **Thermal and geometric properties** in the **Component Parameters** section. The material and geometric properties of the p-type semiconductors should be set in this section.

First, set the **Seebeck coefficient**, S_{p}.

Electrical and Thermal Resistances

Either specify directly the **Electrical resistance**, $R_{e,p}$ and **Thermal resistance**, R_p or specify the **Electrical conductivity**, $\sigma_{e,p}$ and **Thermal conductivity**, k_p to calculate the corresponding resistances.

Thermal Capacitance

Either specify directly the **Thermal capacitance**, C_p or specify the **Density**, ρ_p and **Heat capacity at constant pressure**, $C_{p,p}$ to calculate the corresponding thermal capacitance. For a steady-state problem, heat storage is null and these inputs are ignored.

Geometry Parameters

The **Section area**, A_p and **Length**, L_p of the p-type semiconductors are needed when either the resistances are expressed from the conductivities or the capacitance is expressed from the density and heat capacity at constant pressure.

Material

Select any material from the **Material** list to define some material properties **From material**. For **User defined** enter values or expressions for these properties.

N-TYPE SEMICONDUCTOR PARAMETERS

This section is available when **Specify** is set to **Thermal and geometric properties** in the **Component Parameters** section. The material and geometric properties of the n-type semiconductors should be set in this section. See P-Type Semiconductor Parameters section for details.

CERAMIC PLATE PARAMETERS

This section is available when **Specify** is set to **Thermal and geometric properties** in the **Component Parameters** section. The material and geometric properties of the ceramic plates should be set in this section.

Thermal Capacitance

Either specify directly the **Thermal capacitance**, C_c or specify the **Density**, ρ_c and **Heat capacity at constant pressure**, $C_{p,c}$ to calculate the corresponding thermal capacitance. For a steady-state problem, heat storage is null and these inputs are ignored.

Geometry Parameters

The **Section area**, A_c and **Thickness**, d_c of the ceramic plates are needed when the capacitance is expressed from the density and heat capacity at constant pressure.

Material

Select any material from the **Material** list to define some material properties **From material**. For **User defined** enter values or expressions for these properties.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$, to be used at initialization, in particular to evaluate the material properties, the performance parameters of the thermoelectric module, and the heat storage term.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature at node 1**
- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus
Lumped Thermal System>Thermoelectric Module

Ribbon
Physics tab with **Lumped Thermal System** selected:

Global>Thermoelectric Module

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
- Boundary Features
- Global Features

Domain Features

The Moisture Transport interface has the following domain nodes available:

- Building Material
- Hygroscopic Porous Medium
- Initial Values
- Liquid Water (Hygroscopic Porous Medium)
- Moist Air (Hygroscopic Porous Medium)
- Moist Air (Moisture Transport Interface)
- Moisture Source
- Porous Matrix (Hygroscopic Porous Medium)
- Turbulent Mixing

> For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

Building Material

Use this node to model moisture transfer in a building material through vapor diffusion and capillary moisture flows. The moisture content variation is expressed through the transfer of relative humidity

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot \mathbf{g}_w = G \tag{7-1}$$

$$\mathbf{g}_w = -(\xi D_w \nabla \phi_w + \delta_p \nabla(\phi_w p_{sat})) \tag{7-2}$$

$$\xi = \frac{\partial w(\phi_w)}{\partial \phi_w}$$

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_w (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: $kg/(m^3 \cdot s)$) is the moisture source (or sink). See the Moisture Source node.

For a steady-state problem, the relative humidity does not change with time and the first term disappears.

The transfer equation above, specifically designed for building materials under various assumptions, should be used carefully for some other type of porous media.

MODEL INPUT

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** ( ).

BUILDING MATERIAL

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** $w(\phi_w)$ 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.

$\mathbb{Q}$	The Moisture Transport in Porous Media Interface The Heat and Moisture Transport Interfaces Heat and Moisture Theory for Moisture Transport in Building Materials

📄	See Building Material Library in the *COMSOL Multiphysics Reference Manual* for a description of materials containing the thermal and hygroscopic properties required by the Building Material node.

$\mathbb{Q}$	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

▥	*Heat and Moisture Transport in a Semi-Infinite Wall*: Application Library path **Heat_Transfer_Module/Verification_Examples/ semi_infinite_wall**

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Building Material
Moisture Transport in Building Materials>Building Material
Moisture Transport in Porous Media>Building Material

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Domains>Building Material

Hygroscopic Porous Medium

Use this node to model moisture transport in a porous medium filled with two phases in equilibrium: liquid water and moist air. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

The variation of the total moisture content is expressed through the sum of the transport of vapor in moist air and liquid water. See Theory for Moisture Transport in Porous Media for details.

The liquid saturation, s_l, which describes the amount of liquid water within the pores, is defined from the specification of a moisture storage function $w(\phi_w)$, which specifies the total moisture content (left hand side of the equation below) as a function of the relative humidity ϕ_w :

$$\varepsilon_p s_l \rho_l + \varepsilon_p \rho_g \omega_v (1 - s_l) \ = \ w(\phi_w)$$

where ε_p (dimensionless) is the porosity.

When there is no variation of the total pressure and under specific temperature conditions, the Building Material node may be used instead. See Theory for Moisture Transport in Building Materials for details.

MODEL INPUT

This section has fields and values that are inputs to expressions that define material properties in the **Liquid Water**, **Moist Air**, and **Porous Matrix** subnodes.

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 Moist Porous Media** 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** (![icon]).

COORDINATE SYSTEM SELECTION

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate system**, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The **Hygroscopic Porous Medium** node and its subnodes inherit these coordinate system settings. In particular, the **Moist air velocity field** (in Moist Air (Hygroscopic Porous Medium) subnode), the **Permeability** (in Porous Matrix (Hygroscopic Porous Medium) subnode), and the **Capillary liquid flux density** (in Hygroscopic Porous Medium node) should be set according to the coordinate system selected in this section.

See Coordinate Systems in the *COMSOL Multiphysics Reference Manual* for more details.

MOISTURE TRANSPORT PROPERTIES

The **Moisture storage function** $w(\phi_w)$ 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.

Then, the following options are available for the capillary flux:

- When **Capillary model** is set to **Kelvin's law** (default), a Darcy's Law based on the capillary pressure is used, and the capillary pressure itself is expressed from the relative humidity by using Kelvin's law, using the assumption that the liquid and gas phases are in equilibrium in the porous medium.

- When **Capillary model** is set to **Diffusion model**, the capillary flux is expressed from the gradient of the moisture content, and the **Moisture diffusivity** D_w, should be specified.

- For **User defined**, set a value for the **Capillary liquid flux density**, $\mathbf{g}_{lc}$, to characterize the liquid transport in function of the moisture content.

When a **Porous Material** node is defined on the domains where the **Hygroscopic Porous Medium** feature is active, the **Moisture storage function** and the **Moisture diffusivity** can be set in the **Homogenized Properties** section of the **Porous Material** node.

| | See Porous Material in the *COMSOL Multiphysics Reference Manual.* |

	The moist air properties can be specified in the Moist Air (Hygroscopic Porous Medium) subnode.
	The properties for liquid water convective transport can be specified in the Liquid Water (Hygroscopic Porous Medium) subnode.
	The solid matrix porosity and permeability can be specified in the Porous Matrix (Hygroscopic Porous Medium) subnode.

	• The Moisture Transport in Porous Media Interface
	• The Heat and Moisture Transport Interfaces
	• Heat and Moisture
	• Theory for Moisture Transport in Porous Media

| | For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/ |

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Hygroscopic Porous Medium
Moisture Transport in Building Materials>Hygroscopic Porous Medium
Moisture Transport in Porous Media>Hygroscopic Porous Medium

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Domains>Hygroscopic Porous Medium

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_w. The default value is 0.5. Else, select an **Ambient relative humidity** defined in an **Ambient Properties** node under **Definitions**.

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Initial Values
Moisture Transport in Building Materials>Initial Values
Moisture Transport in Porous Media>Initial Values

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Domains>Initial Values

Liquid Water (Hygroscopic Porous Medium)

This subnode defines the properties for the transport of the liquid water phase in a porous medium filled with two phases in equilibrium: liquid water and moist air. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

LIQUID WATER PROPERTIES

This section contains settings for the definition of convective transport of liquid water. See Hygroscopic Porous Medium for the definition of capillary transport.

Set the **Relative liquid water permeability**, κ_{rl} (dimensionless), that multiplies the porous medium permeability in the Darcy's Law to account for the presence of the liquid phase.

The convective flux of liquid water is always computed with Darcy's Law.

⌕	The liquid water properties are based on the temperature set in the corresponding model input in the Hygroscopic Porous Medium parent node.
	The capillary transport and storage properties can be specified in the Hygroscopic Porous Medium parent node.
	The liquid saturation, s_l, is defined from the moisture storage function specified in the Hygroscopic Porous Medium parent node.
	The moist air properties can be specified in the Moist Air (Hygroscopic Porous Medium) subnode.
	The solid matrix porosity and permeability can be specified in the Porous Matrix (Hygroscopic Porous Medium) subnode.

⌕	The Moisture Transport in Porous Media Interface
	The Heat and Moisture Transport Interfaces
	Heat and Moisture
	Theory for Moisture Transport in Porous Media

⌕	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air>Hygroscopic Porous Medium>Liquid Water
Moisture Transport in Building Materials>Hygroscopic Porous Medium>Liquid Water
Moisture Transport in Porous Media>Hygroscopic Porous Medium>Liquid Water

Ribbon
Physics Tab with **Hygroscopic Porous Medium** selected in the model tree:

Attributes>Liquid Water

Moist Air (Hygroscopic Porous Medium)

This subnode defines the properties for the transport of the moist air phase in a porous medium filled with two phases in equilibrium: liquid water and moist air. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

MOIST AIR PROPERTIES

This section contains settings for the definition of convective and diffusive vapor transport in the moist air phase.

First, set the **Diffusion Coefficient**, D (SI unit: m^2/s), for the binary diffusion of vapor in air in a free medium. The default value $2.6 \cdot 10^{-5}$ m²/s is valid at $T = 298$ K and $p_A = 1$ atm. You may set a temperature or pressure-dependent diffusion coefficient through the use of the variables `mt.T` and `mt.pA`.

Specify the **Effective diffusivity model** used to account for the porosity and the tortuosity of the porous medium in the diffusion coefficient, D_{eff} (SI unit: m^2/s). The available options are **Millington and Quirk model** (the default), **Bruggeman model**, **Tortuosity model**, and **No correction**.

For the **Tortuosity model** option, enter a value for the **Tortuosity factor**, τ (dimensionless). The default is 1.

Then, set the **Moist air velocity field**, $\mathbf{u}_g$, that should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area. 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).

⊕	When setting temperature or pressure-dependent properties, the corresponding model inputs should be specified in the Hygroscopic Porous Medium parent node.
	The capillary transport and storage properties can be specified in the Hygroscopic Porous Medium parent node.
	The liquid water properties can be specified in the Liquid Water (Hygroscopic Porous Medium) subnode.
	The solid matrix porosity and permeability can be specified in the Porous Matrix (Hygroscopic Porous Medium) subnode.

For a detailed overview of the functionality available in each product, visit
https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Hygroscopic Porous Medium>Moist Air
Moisture Transport in Building Materials>Hygroscopic Porous Medium>Moist Air
Moisture Transport in Porous Media>Hygroscopic Porous Medium>Moist Air

Ribbon
Physics Tab with **Hygroscopic Porous Medium** selected in the model tree:

Attributes>Moist Air

Moist Air (Moisture Transport Interface)

Use this node to model moisture transfer in moist air through vapor diffusion and
convection.

For small vapor concentration conditions, the density of moist air is supposed not to
change with variations of moisture content, and a diluted species formulation is used.
In this case, the moisture content variation is expressed through the transport of vapor
concentration c_v:

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w = G \qquad (7\text{-}3)$$

$$\mathbf{g}_w = -M_v D \nabla c_v$$

$$c_v = \phi_w c_{sat}$$

with the following material properties, fields, and source:

- M_v (SI unit: kg/mol) is the molar mass of water vapor.

- ϕ_w (dimensionless) is the relative humidity.

- c_{sat} (SI unit: mol/m^3) is the vapor saturation concentration.

- D (SI unit: m^2/s) is the vapor diffusion coefficient in air.

- $\mathbf{u}$ (SI unit: m/s) is the air velocity field.

- G (SI unit: kg/(m^3·s)) is the moisture source (or sink). See Moisture Source node.

For larger vapor concentration conditions, the variation of moist air density are modeled, and a concentrated species formulation is used. In this case, the moisture content variation is expressed through the transport of vapor mass fraction ω_v:

$$\rho_g \frac{\partial \omega_v}{\partial t} + \rho_g \mathbf{u} \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w = G \tag{7-4}$$

$$\mathbf{g}_w = -\rho_g D \nabla \omega_v$$

$$\omega_v = \frac{M_v c_v}{\rho_g}$$

$$c_v = \phi_w c_{sat}$$

where ρ_g is the moist air density (SI unit: kg/m^3).

For a steady-state problem, the relative humidity does not change with time and the first term in Equation 7-3 and Equation 7-4 disappears.

MODEL INPUT

This section has fields and values that are inputs to expressions used in Equation 7-3 and Equation 7-4. The temperature is used for the definition of the saturation conditions, whereas the velocity field is used in the convection term.

The default **Temperature** T and **Velocity field u** are **User defined**. When additional physics interfaces are added to the model, the temperature and velocity field 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 **Velocity field (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** ( ).

DIFFUSION

Set a value for the **Diffusion coefficient**, D. The default value 2.6e-5 m²/s is valid at T=298 K and pA=1 atm. You may set a temperature or pressure-dependent diffusion coefficient through the use of the variables `mt.T` and `mt.pA`.

The Moisture Transport in Air Interface

The Heat and Moisture Transport Interfaces

Heat and Moisture

Theory for Moisture Transport in Air

See Turbulent Mixing to account for eddy diffusivity in moisture convection by turbulent flows.

For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

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**

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Moist Air
Moisture Transport in Building Materials>Moist Air
Moisture Transport in Porous Media>Moist Air

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Domains>Moist Air

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 in Air>Moisture Source
Moisture Transport in Building Materials>Moisture Source
Moisture Transport in Porous Media>Moisture Source

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Domains>Moisture Source

Porous Matrix (Hygroscopic Porous Medium)

This subnode defines the properties of a porous medium filled with two phases in equilibrium: liquid water and moist air. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

POROUS MATRIX PROPERTIES

First, set the **Porosity**, ε_p (a dimensionless number between 0 and 1), that uses by default the value **From material**. For **User defined** the default is 0.

Then, set the **Permeability**, κ (SI unit: m^2), to specify the capacity of the porous material to transmit flow. For **User defined** select **Isotropic** to define a scalar value or **Diagonal**, **Symmetric**, or **Full** to define a tensor value and enter another value or expression in the field or matrix.

When a **Porous Material** node is defined on the domains where the **Hygroscopic Porous Medium** feature is active, the **Porosity** and the **Permeability** can be set in the **Homogenized Properties** section of the **Porous Material** node.

ℚ	When setting temperature or pressure-dependent properties, the corresponding model inputs can be specified in the Hygroscopic Porous Medium parent node.
	The capillary transport and storage properties of the porous material can be specified in the Hygroscopic Porous Medium parent node.
	The properties for convective liquid transport can be specified in the Liquid Water (Hygroscopic Porous Medium) subnode.
	The moist air properties can be specified in the Moist Air (Hygroscopic Porous Medium) subnode.

ℚ	The Moisture Transport in Porous Media Interface
	The Heat and Moisture Transport Interfaces
	Heat and Moisture
	Theory for Moisture Transport in Porous Media

ℚ	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Hygroscopic Porous Medium>Porous Matrix
Moisture Transport in Building Materials>Hygroscopic Porous Medium>Porous Matrix
Moisture Transport in Porous Media>Hygroscopic Porous Medium>Porous Matrix

Physics Tab with **Hygroscopic Porous Medium** selected in the model tree:

Attributes>Porous Matrix

Turbulent Mixing

This subnode should be used to account for the turbulent mixing caused by the eddy diffusivity in moisture convection, for example when the specified velocity field corresponds to a RANS solution.

The **Turbulent Mixing** node adds the following contribution to the diffusion coefficient *D* of Equation 7-3 and Equation 7-4 of Moist Air (Moisture Transport Interface) node:

$$D_{\mathrm{T}} = \frac{\nu_{\mathrm{T}}}{Sc_{\mathrm{T}}}$$

where ν_{T} is the turbulent kinematic viscosity (SI unit: m^2/s) and Sc_{T} is the turbulent Schmidt number (dimensionless).

TURBULENT MIXING PARAMETERS

Enter a value or expression for the **Turbulent kinematic viscosity** ν_{T} and the **Turbulent Schmidt number** Sc_{T}. If a turbulent fluid flow interface is active, the **Turbulent kinematic viscosity** can be taken directly from this interface.

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Moist Air>Turbulent Mixing
Moisture Transport in Building Materials>Moist Air>Turbulent Mixing
Moisture Transport in Porous Media>Moist Air>Turbulent Mixing

Ribbon
Physics Tab with **Moist Air** selected in the model tree:

Domains>Turbulent Mixing

Boundary Features

The Moisture Transport interface has the following boundary nodes available:

- Continuity
- Inflow
- Insulation
- Moist Surface
- Moisture Content
- Moisture Flux

- Open Boundary
- Outflow
- Periodic Condition
- Symmetry
- Thin Moisture Barrier
- Wet Surface

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 https://www.comsol.com/products/specifications/

Continuity

This node can be added to pairs. It prescribes that the moisture content is continuous across the pair. **Continuity** is only suitable for pairs where the boundaries match. Depending on the domain conditions active in the adjacent domains, the constraint is applied either on relative humidity, vapor concentration or vapor mass fraction on boundary pairs. For continuous temperature and pressure conditions, all these constraints are equivalent.

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.

To display this section, click the **Show More Options** button ( ) and select **Advanced Physics Options**.

The **Discontinuous Galerkin constraints** option ensures continuity of the diffusive flux. This option may work better than the **Classical constraints** one when the meshes of the source and destination boundaries do not coincide.

Alternatively, select the **Classical constraints** option. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

ADVANCED

Select the **Disconnect pair** check box to disable the continuity condition.

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Pairs>Continuity
Moisture Transport in Building Materials>Pairs>Continuity
Moisture Transport in Porous Media>Pairs>Continuity

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Pairs>Continuity

Inflow

Use this node to model inflow of moisture through a virtual domain, at inlet boundaries of moist air domains. The upstream moisture conditions (either vapor concentration c_{ustr}, or vapor mass fraction ω_{ustr}, or relative humidity $\phi_{w, ustr}$ and temperature T_{ustr}) are known at the outer boundary of the virtual domain. The inflow condition is often similar to a moisture content condition. However using this condition the upstream vapor concentration is not strictly enforced at the inlet. The higher the flow rate, the smallest difference between the inlet and upstream moisture content. In case of moisture sources or moisture constraints close to the inlet boundary, the inflow condition induces a moisture profile that approaches to the moisture profile that would be obtained by computing the solution in the virtual domain upstream the inlet.

This boundary condition estimates the moisture flux through the inlet boundary from a Danckwerts condition on the concentration:

$$-\mathbf{n} \cdot \mathbf{g}_{\mathrm{w}} = M_{\mathrm{v}}(c_{\mathrm{v}} - c_{\mathrm{ustr}})\mathbf{u} \cdot \mathbf{n}$$

When the **Concentrated species** option is selected in the **Mixture type for moist air** list, this condition is rewritten as:

$$-\mathbf{n} \cdot \mathbf{g}_{\mathrm{w}} = \rho_{\mathrm{g}}(\omega_{\mathrm{v}} - \omega_{\mathrm{ustr}})\mathbf{u} \cdot \mathbf{n}$$

to be expressed in function of the mass fraction ω_{v}, with

$$\omega_{\mathrm{ustr}} = \frac{M_{\mathrm{v}}c_{\mathrm{ustr}}}{\rho_{\mathrm{g}}}$$

This node can be applied on the exterior boundaries of a computational domain in which a Moist Air (Moisture Transport Interface) node is active.

UPSTREAM PROPERTIES

When **Specify** is set to **Relative humidity** in the **Specify** list, set both the **Upstream temperature** T_{ustr} and the **Upstream relative humidity** $\phi_{\mathrm{w, ustr}}$ from which the upstream vapor concentration can be defined:

$$c_{\mathrm{ustr}} = \phi_{\mathrm{w, ustr}}c_{\mathrm{sat}}(T_{\mathrm{ustr}})$$

For **User Defined**, enter values or expressions. Else, select any available input (like **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**).

Alternatively, choose **Vapor concentration** in the **Specify** list, and set directly a value or expression for the **Upstream vapor concentration** c_{ustr}.

Finally, choose **Vapor mass fraction** in the **Specify** list to set a value or expression for the **Upstream vapor mass fraction** ω_{ustr}.

| ▥ | *Evaporative Cooling of Water*: Application Library path **Heat_Transfer_Module/Phase_Change/evaporative_cooling** |

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Flow Conditions>Inflow
Moisture Transport in Building Materials>Flow Conditions>Inflow

Moisture Transport in Porous Media>Flow Conditions>Inflow

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:

$$-\mathbf{n} \cdot \mathbf{g}_{\mathrm{w}} = 0$$

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

Moist Surface

This node should be used to model evaporation from and condensation to a solid surface partially covered with liquid water. It adds the evaporation flux to the boundary conditions of Equation 7-3 and Equation 7-4 of Moist Air (Moisture Transport Interface) node, and computes the corresponding latent heat source to be added in the heat transfer equation. In addition, it computes the liquid water concentration accumulated on the surface, during evaporation and condensation processes.

It can be applied on the exterior boundaries of a domain where the Moist Air (Moisture Transport Interface) node is active.

In time-dependent studies, the evaporation flux g_{evap} (SI unit: kg/(m^2·s)) is defined as:

$$g_{evap} = \begin{cases} M_v K(c_{sat} - c_v) \text{ if } c_v > c_{sat} \text{or } c_l > 0 \\ 0 \text{ otherwise} \end{cases}$$

where M_v is the molar mass of water vapor (SI unit: kg/mol), K is the evaporation rate factor (SI unit: m/s), c_{sat} is the saturation concentration of vapor (SI unit: mol/m^3), c_v is the vapor concentration (SI unit: mol/m^3), and c_l is the liquid water concentration on surface (SI unit: mol/m^2).

The liquid water concentration on surface is computed by solving the following equation:

$$M_v \frac{\partial c_l}{\partial t} = -g_{evap}$$

with the initial condition:

$$c_l(0) = c_{l, init}$$

In stationary studies, the initial liquid concentration is used in the definition of the evaporation flux:

$$g_{evap} = \begin{cases} M_v K(c_{sat} - c_v) \text{ if } c_v > c_{sat} \text{or } c_{l, init} > 0 \\ 0 \text{ otherwise} \end{cases}$$

The latent heat source q_{evap} (SI unit: W/m^2) is obtained by multiplying the evaporation flux by the latent heat of evaporation L_v (SI unit: J/kg):

$$q_{evap} = L_v g_{evap}$$

With these definitions, g_{evap} is negative when condensation occurs, and positive during evaporation. Whereas the condensation process does not depend on the liquid concentration on the surface, the evaporation process only happens when the (initial or time-dependent) liquid concentration on surface is positive.

MOIST SURFACE SETTINGS

Enter values or expressions for the **Evaporation rate factor** K and the **Initial liquid water concentration on moist surface** $c_{l,init}$.

ⓠ	See Wet Surface node for the modeling of evaporation and condensation on a solid surface completely covered with liquid water.

ⓠ	When a Heat and Moisture multiphysics coupling node is active, the resulting latent heat source is automatically added as a boundary heat source into the heat transfer equation.

▥	*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**

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air>Moist Surface
Moisture Transport in Building Materials>Moist Surface
Moisture Transport in Porous Media>Moist Surface

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Moist Surface

Moisture Content

Use this node to specify the moisture conditions (relative humidity and temperature, vapor concentration, or vapor mass fraction) on 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.

When **Specify** is set to **Relative humidity**, the equation for this condition is

$$c_v = \phi_{w,0} c_{sat}(T_0)$$

where $\phi_{w,0}$ and T_0 are the prescribed relative humidity and temperature on the boundary. For **User Defined**, enter values or expressions. Else, select any available input (like **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**). If the **Concentrated species** formulation is used, the vapor mass fraction is then defined as:

$$\omega_v = \frac{M_v \phi_{w,0} c_{sat}(T_0)}{M_v \phi_{w,0} c_{sat}(T_0) + M_a \left(\frac{p_A}{RT} - \phi_{w,0} c_{sat}(T_0) \right)}$$

Alternatively, choose **Vapor concentration** in the **Specify** list and set directly the **Vapor concentration** c_0. If the **Concentrated species** formulation is used, the vapor mass fraction is then defined as:

$$\omega_v = \frac{c_0}{M_v c_0 + M_a \left(\frac{p_A}{RT} - c_0 \right)}$$

Finally, choose **Vapor mass fraction** in the **Specify** list to set directly ω_0. If the **Diluted species** formulation is used, the vapor concentration is then defined as:

$$c_v = \frac{M_a \omega_0}{M_v (1 - \omega_0) + M_a \omega_0} \cdot \frac{p_A}{RT}$$

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

The **Discontinuous Galerkin constraints** option ensures continuity of the diffusive flux.

▥	*Heat and Moisture Transport in a Semi-Infinite Wall*: Application Library path **Heat_Transfer_Module/Verification_Examples/ semi_infinite_wall**

Context Menus
Moisture Transport in Air>Moisture Content
Moisture Transport in Building Materials>Moisture Content
Moisture Transport in Porous Media>Moisture Content
Moisture Transport in Air>Pairs>Moisture Content
Moisture Transport in Building Materials>Pairs>Moisture Content
Moisture Transport in Porous Media>Pairs>Moisture Content

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Moisture Content
Pairs>Moisture Content

Moisture Flux

Use this node to add moisture flux across exterior boundaries, with the option to account for latent heat source due to evaporation. A positive moisture flux adds moisture to the domain.

MOISTURE FLUX

Select the **Flux type** from the list: **General moisture flux** (the default), **Convective moisture flux, concentrations difference**, or **Convective moisture flux, pressures difference**.

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, Concentrations Difference
Several options are available to specify the **Moisture transfer coefficient** h_m.

- The default option is to enter a **User defined** value for h_m.
- With the **From heat and moisture transport analogy** option, the heat transfer coefficient h to be multiplied by the analogy factor to obtain the moisture transfer coefficient h_m should be specified. It can be **User defined** (in this case the **Absolute pressure**, p_A, the **External temperature**, T_{ext}, and the **External relative humidity**, $\phi_{w,\,ext}$ should be specified) or selected from a list of heat transfer coefficients

defined in Heat Flux features applied on the same boundary (in this case the external conditions defined in the corresponding **Heat Flux** feature are used).

- For the **External natural convection**, **Internal natural convection**, **External forced convection**, and **Internal forced convection** options, the geometry and velocity parameters used in the correlation are the same as for the Heat Flux feature. Then the external conditions should be specified. First, set the **Absolute pressure**, p_A. For **User Defined**, enter a value or expression. Else, select an **Ambient absolute pressure** defined in an **Ambient Properties** node under **Definitions**. In addition, enter an **External temperature**, T_{ext}. For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. Finally, set the **External relative humidity**, $\phi_{w,\,ext}$. Note that the relative humidity at the boundary is provided by the physics interface.

For all options except **User defined**, select the **Specify the exponent in the analogy factor** check box to set the variable n in Equation 4-191.

Convective Moisture Flux, Pressures Difference

Enter a value for the **Moisture transfer coefficient** β_p. In addition, two options are available to specify the external conditions:

- If **Relative humidity** is selected (the default), set the **External relative humidity**, $\phi_{w,\,ext}$ 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 an **Ambient Properties** node under **Definitions**. Convective moisture flux is defined by $g_0 = \beta_p(\phi_{w,\,ext}\,p_{sat}(T_{ext}) - \phi_w\,p_{sat}(T))$.
- If **Partial vapor pressure** is selected, enter an **External partial vapor pressure**, $p_{v,ext}$. Convective moisture flux is defined by $g_0 = \beta_p(p_{v,ext} - \phi_w\,p_{sat}(T))$.

EVAPORATION

Select the **Contributes to evaporation flux** check box to define the moisture flux as an evaporation flux, and to contribute to the total latent heat source mt.q_evaptot.

▥	*Condensation Risk in a Wood-Frame Wall*: Application Library path **Heat_Transfer_Module/Buildings_and_Constructions/wood_frame_wall**

Context Menus

Moisture Transport in Air>Moisture Flux
Moisture Transport in Building Materials>Moisture Flux
Moisture Transport in Porous Media>Moisture Flux

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Moisture Flux

Open Boundary

Use this condition to define a boundary as the limit between a moist air domain of the geometry and the rest of the same moist air domain that is not represented in the geometry.

This node can be applied on the exterior boundaries of a computational domain in which a Moist Air (Moisture Transport Interface) node is active.

At the open boundary both inflow and outflow conditions are supported.

For an outgoing moist airflow across the boundary, it applies a zero diffusive flux condition:

$$-\mathbf{n} \cdot \mathbf{g}_w = 0, \qquad \mathbf{n} \cdot \mathbf{u} \geq 0 \tag{7-5}$$

For an incoming flow of velocity field **u** across the boundary, it accounts by default for the moisture flux induced by the flow rate through a Danckwerts condition on the vapor concentration:

$$-\mathbf{n} \cdot \mathbf{g}_w = M_v(c_v - c_{ustr})\mathbf{u} \cdot \mathbf{n}, \qquad \mathbf{n} \cdot \mathbf{u} < 0$$

When the **Concentrated species** option is selected in the **Mixture type for moist air** list, this condition is rewritten as:

$$-\mathbf{n} \cdot \mathbf{g}_w = \rho_g(\omega_v - \omega_{ustr})\mathbf{u} \cdot \mathbf{n}, \qquad \mathbf{n} \cdot \mathbf{u} < 0$$

to be expressed in function of the mass fraction ω_v, with

$$\omega_{ustr} = \frac{M_v c_{ustr}}{\rho_g}$$

Alternatively the open boundary condition can set a constraint on the vapor concentration or mass fraction for an incoming flow, depending on the option is selected in the **Mixture type for moist air** list:

$$c_v = c_{ustr}, \qquad \mathbf{n} \cdot \mathbf{u} < 0 \qquad\qquad (7\text{-}6)$$

$$\omega_v = \omega_{ustr}, \qquad \mathbf{n} \cdot \mathbf{u} < 0 \qquad\qquad (7\text{-}7)$$

Both types of conditions are based on the knowledge of the upstream moisture content (from the vapor concentration, the vapor mass fraction, or the relative humidity and the temperature) at the outer boundary of the virtual domain, but the concentration and mass fraction ones should be preferred to account for the feedback of the model moisture sources and moisture constraints on the inlet moisture profile.

UPSTREAM PROPERTIES

When **Specify** is set to **Relative humidity** in the **Specify** list, set both the **Upstream temperature** T_{ustr} and the **Upstream relative humidity** $\phi_{w, ustr}$ from which the upstream vapor concentration can be defined:

$$c_{ustr} = \phi_{w, ustr} c_{sat}(T_{ustr})$$

For **User Defined**, enter values or expressions. Else, select any available input (like **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**).

Alternatively, choose **Vapor concentration** in the **Specify** list, and set directly a value or expression for the **Upstream vapor concentration** c_{ustr}.

Finally, choose **Vapor mass fraction** in the **Specify** list to set a value or expression for the **Upstream vapor mass fraction** ω_{ustr}.

INFLOW CONDITION

Unfold this section to change the inflow condition:

• The **Flux (Danckwerts)** condition (default) prescribes the flux defined from the upstream vapor concentration (or mass fraction) and the fluid velocity at the boundary. Using this condition, the moisture condition is not strictly enforced at

the inlet. Instead a flux is set on this boundary. The higher the flow rate, the smallest difference between the inlet and upstream vapor concentration (or mass fraction).

- The **Discontinuous Galerkin constraints** option enforces the upstream vapor concentration (for **Diluted species** option) or the upstream vapor mass fraction (for **Concentrated species** option) on the boundary.

<div style="border:1px solid; padding:8px;">

▥ *Evaporative Cooling of Water*: Application Library path
Heat_Transfer_Module/Phase_Change/evaporative_cooling

</div>

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Flow Conditions>Open Boundary
Moisture Transport in Building Materials>Flow Conditions>Open Boundary
Moisture Transport in Porous Media>Flow Conditions>Open Boundary

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Open Boundary

Outflow

This node provides a boundary condition at outlets where the vapor is transported out of the air domain. It is assumed that convection is the dominating transport mechanism across outflow boundaries, and therefore that diffusive transport can be ignored, that is:

$$-\mathbf{n} \cdot D\nabla(M_v \phi_w c_{sat}) = 0$$

This node can be applied on the exterior boundaries of a computational domain in which a Moist Air (Moisture Transport Interface) node or an Hygroscopic Porous Medium node is active.

LOCATION IN USER INTERFACE

Context Menus
Moisture Transport in Air>Flow Conditions>Outflow
Moisture Transport in Building Materials>Flow Conditions>Outflow

Moisture Transport in Porous Media>Flow Conditions>Outflow

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Outflow

Periodic Condition

Use this node to prescribe periodic conditions on moisture. Depending on the domain condition active in the adjacent domain, the constraint is applied either on relative humidity, vapor concentration or vapor mass fraction on boundary pairs. For periodic temperature and pressure conditions, all these constraints are equivalent.

BOUNDARY SELECTION

The software automatically identifies the boundaries as either source boundaries or destination boundaries. The selection for the destination can be changed by right-clicking the **Periodic Condition** node to add a **Destination Selection** subnode.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

The **Discontinuous Galerkin constraints** option ensures continuity of the diffusive flux. This option may work better than the **Classical constraints** one when the meshes of the source and destination boundaries do not coincide.

Alternatively, select the **Classical constraints** option. Select the **Use weak constraints** check box to replace the standard constraints with a weak implementation.

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

See Orientation of Source and Destination in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button (👁) and select **Advanced Physics Options**.

See Mapping Between Source and Destination in the *COMSOL Multiphysics Reference Manual*.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air>Periodic Condition
Moisture Transport in Building Materials>Periodic Condition
Moisture Transport in Porous Media>Periodic Condition

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Periodic Condition

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 in Air>Symmetry
Moisture Transport in Building Materials>Symmetry
Moisture Transport in Porous Media>Symmetry

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Symmetry

Thin Moisture Barrier

Use this node to model a discontinuous moisture content across interior boundaries. By default it models a vapor barrier, with the moisture transfer coefficient β accounting for vapor diffusion only. A more general moisture barrier can be defined through the specification of a user-defined moisture transfer coefficient.

The upside and downside moisture fluxes are defined by $\beta(\phi_{w,d} - \phi_{w,u})$ and $\beta(\phi_{w,u} - \phi_{w,d})$, respectively.

MODEL INPUT

This section has fields and values that are inputs to expressions that define material properties for vapor permeability, vapor resistance factor, and saturation pressure of water vapor. 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 **Vapor barrier material properties** is selected (the default), the moisture transfer coefficient β is defined as

$$\beta = \frac{\delta p_{sat}}{\mu d_s} = \frac{\delta_p p_{sat}}{d_s} = \frac{\delta p_{sat}}{s_d}$$

where δ is the vapor permeability of still air (SI unit: s), p_{sat} is the saturation pressure of water vapor (SI unit: Pa), μ is the vapor resistance factor (dimensionless), δ_p is the vapor permeability (SI unit: s), s_d is the vapor diffusion equivalent air layer thickness (SI unit: m), and d_s is the layer thickness (SI unit: m).

Depending on the option selected in **Specify** — **Vapor resistance factor** (the default), **Vapor permeability**, or **Vapor diffusion equivalent air layer thickness**, enter values or expressions for the needed properties, or use properties **From material**.

• If **Moisture transfer coefficient** is selected, enter a value or expression for the **Moisture transfer coefficient** β.

	See Building Material Library in the *COMSOL Multiphysics Reference Manual* for a description of materials containing the thermal and hygroscopic properties required by the Thin Moisture Barrier node.

	Condensation Risk in a Wood-Frame Wall: Application Library path **Heat_Transfer_Module/Buildings_and_Constructions/wood_frame_wall**

Context Menus
Moisture Transport in Air>Thin Moisture Barrier
Moisture Transport in Building Materials>Thin Moisture Barrier
Moisture Transport in Porous Media>Thin Moisture Barrier

Ribbon
Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Thin Moisture Barrier

Wet Surface

This node should be used to model evaporation from and condensation to a solid surface completely covered with liquid water. It adds the evaporation flux to the boundary conditions of Equation 7-3 or Equation 7-4 of Moist Air (Moisture Transport Interface) node, and computes the corresponding latent heat source to be added in the heat transfer equation.

It can be applied on the exterior boundaries of a domain where the Moist Air (Moisture Transport Interface) node is active.

The evaporation flux g_{evap} (SI unit: kg/(m^2·s)) is defined as:

$$g_{evap} = M_v K(c_{sat} - c_v)$$

where M_v is the molar mass of water vapor (SI unit: kg/mol), K is the evaporation rate factor (SI unit: m/s), c_{sat} is the saturation concentration of vapor (SI unit: mol/m^3), and c_v is the vapor concentration (SI unit: mol/m^3).

The latent heat source q_{evap} (SI unit: W/m^2) is obtained by multiplying the evaporation flux by the latent heat of evaporation L_v (SI unit: J/kg):

$$q_{evap} = L_v g_{evap}$$

WET SURFACE SETTINGS

Enter a value or expression for the **Evaporation rate factor** K.

?	See Moist Surface node for the computation of the amount of liquid water accumulated on a solid surface partially covered with liquid water, due to evaporation and condensation.

?	When a Heat and Moisture multiphysics coupling node is active, the resulting latent heat source is automatically added as a boundary heat source into the heat transfer equation.

▥	*Evaporative Cooling of Water*: Application Library path **Heat_Transfer_Module/Phase_Change/evaporative_cooling**

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air>Wet Surface
Moisture Transport in Building Materials>Wet Surface
Moisture Transport in Porous Media>Wet Surface

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Wet Surface

Global Features

The Moisture Transport interface has the following global node available:

- Gravity

> For a detailed overview of the functionality available in each product, visit
> https://www.comsol.com/products/specifications/

Gravity

This global feature is automatically added when the **Account for gravity in liquid velocity** check box is selected at the interface level in the **Physical Model** section. It defines the gravity forces from the **Acceleration of gravity** value, **g**, for the computation of the liquid water flux, $\mathbf{g}_l$:

$$
\mathbf{g}_l = -\frac{\kappa_{rl}\kappa}{\mu_l}(\nabla p_A - \rho_l \mathbf{g})\rho_l + \mathbf{g}_{lc}
$$

See Hygroscopic Porous Medium for details about the other terms in the equation above.

ACCELERATION OF GRAVITY

The **Acceleration of gravity** (SI unit m/s) should be set to a global quantity. It is set by default to $-g_{const}\mathbf{e}_z$ in 2D axisymmetric and 3D models, and $-g_{const}\mathbf{e}_y$ in 2D models.

8

Multiphysics Interfaces

The Heat Transfer Module includes predefined multiphysics interfaces for conjugate heat transfer, radiative heat transfer, thermoelectric effect, coupled heat transfer and moisture transport, and moisture flow modeling.

Some of these interfaces couple an interface of the Heat Transfer Module with an interface of another module (the CFD Module or 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 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 Nonisothermal Flow and Conjugate Heat Transfer Interfaces
• The Heat Transfer with Surface-to-Surface Radiation Interface

- The Heat Transfer with Radiation in Participating Media Interface
- The Heat Transfer with Radiation in Absorbing-Scattering Media Interface
- The Heat Transfer with Radiative Beam in Absorbing Media Interface
- The Thermoelectric Effect Interface
- The Heat and Moisture Transport Interfaces
- The Moisture Flow Interfaces
- The Heat and Moisture Flow Interfaces

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, Solid 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 and The Moisture Transport 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.

In the figure below, the predefined interface **Conjugate Heat Transfer, Laminar Flow** is found under the **Heat Transfer** branch. It adds the Heat Transfer in Solids and Fluids and Laminar Flow constituent interfaces, and the **Multiphysics** node containing the **Nonisothermal Flow** multiphysics coupling.
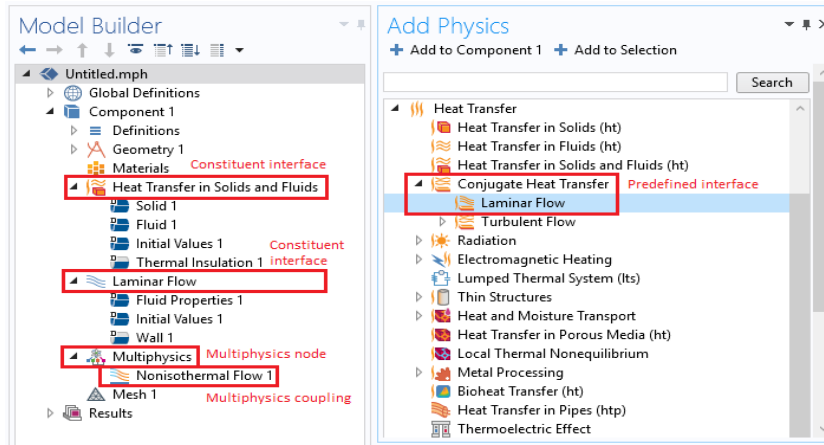
The Nonisothermal Flow and Conjugate Heat Transfer Interfaces

In this section:

- Advantages of Using the Multiphysics Interfaces
- The Nonisothermal 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 Feature
- Physics Interface Features
- Preset Studies

See The Laminar Flow Interface, The Turbulent Flow, Algebraic yPlus Interface, The Turbulent Flow, L-VEL Interface, The Turbulent Flow, k-ε Interface, and The Turbulent Flow, Low Re k-ε 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 Nonisothermal 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 **Moist Air** 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 **Nonisothermal Flow** coupling follows the Boussinesq approximation and evaluates the thermal material properties at the reference temperature define in the heat transfer interface. Hence, the **Phase Change Material** domain model — which requires the temperature dependency of the heat capacity — should be added under **Fluid** only with the **Weakly compressible flow** and **Compressible flow (Ma<0.3)** options.

> 💡 When the **Nonisothermal 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 Nonisothermal Flow, Laminar Flow and Turbulent Flow Interfaces

When a multiphysics interface is added from the **Fluid Flow>Nonisothermal 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 **Nonisothermal Flow**.

> 🔍 The Multiphysics Branch 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, k-ω** interface (≋) combines a Heat Transfer in Fluids interface with a Turbulent Flow, k-ω 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 **Turbulent Flow, SST** interface (≋) combines a Heat Transfer in Fluids interface with a Turbulent Flow, SST interface.

- The **Turbulent Flow, Spalart-Allmaras** interface (≋) combines a Heat Transfer in Fluids interface with a Turbulent Flow, Spalart-Allmaras 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 **Nonisothermal Flow**.

🔍	The Multiphysics Branch 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 in Solids and Fluids interface with a Turbulent Flow, Algebraic yPlus interface.

- The **Turbulent Flow, L-VEL** interface (≋) combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, L-VEL interface.

- The **Turbulent Flow, k-ε** interface (≋) combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, k-ε interface.

- The **Turbulent Flow, k-ω** interface ( ) combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, k-ω interface.

- The **Turbulent Flow, Low Re k-ε** interface ( ) combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, Low Re k-ε interface.

- The **Turbulent Flow, SST** interface ( ) combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, SST interface.

- The **Turbulent Flow, Spalart-Allmaras** interface ( ) combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, Spalart-Allmaras 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 may not be automatically included.

For example, if single Heat Transfer in Fluids and Laminar Flow interfaces are added, COMSOL Multiphysics adds an empty **Multiphysics** node. When you right-click this node, you can add the **Nonisothermal Flow** coupling feature, but the modified settings are not included. To get all the modified settings presented below automatically, you may instead right-click the component node, select **Multiphysics Couplings**, and add the **Nonisothermal Flow** coupling feature.

TABLE 8-1: MODIFIED SETTINGS FOR THE NONISOTHERMAL 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. The **Absolute pressure**, p_A (**Model Input** section) and the **Velocity field**, **u** (**Heat Convection** section) are automatically set to the variables from the **Nonisothermal Flow** multiphysics coupling feature.
Heat Transfer in Fluids	The **Absolute pressure**, p_A (**Model Input** section) and the **Velocity field**, **u** (**Heat Convection** section) are automatically set to the variables from the **Nonisothermal Flow** multiphysics coupling feature.

TABLE 8-1: MODIFIED SETTINGS FOR THE NONISOTHERMAL FLOW INTERFACES

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Laminar Flow / Turbulent Flow	In the **Fluid Properties** default feature, the **Density**, ρ is automatically set to the variable from the **Nonisothermal Flow** multiphysics coupling feature.
	The **Compressibility** option of the fluid flow interface is set to **Weakly Compressible**.
Nonisothermal Flow	The **Fluid flow** and **Heat transfer** interfaces are preselected.

> Note that these settings may be overridden if another predefined coupling is added.

Coupling Feature

See Nonisothermal Flow for a description of the multiphysics coupling.

> 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 (NONISOTHERMAL FLOW)

The available physics features for The Heat Transfer in Fluids Interface are listed in Feature Nodes for the Heat Transfer Interface. Also see Fluid for details about that feature.

HEAT TRANSFER IN SOLIDS AND FLUIDS (CONJUGATE HEAT TRANSFER)

The available physics features for The Heat Transfer in Solids and Fluids Interface are listed in Feature Nodes for the Heat Transfer Interface. Also see Solid and Fluid for details about these features.

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

See the *CFD Module User's Guide* for a description of the available physics features for The Turbulent Flow, Algebraic yPlus Interface, listed in the sections Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

TURBULENT FLOW, L-VEL

See the *CFD Module User's Guide* for a description of the available physics features for The Turbulent Flow, L-VEL Interface, listed in the sections Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

TURBULENT FLOW, k-ε

See the *CFD Module User's Guide* for a description of the available physics features for The Turbulent Flow, k-ε Interface, listed in the sections Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

TURBULENT FLOW, LOW RE k-ε

See the *CFD Module User's Guide* for a description of the available physics features for The Turbulent Flow, Low Re k-ε Interface, listed in the sections Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

Preset Studies

The Stationary, One-Way NITF and Time Dependent, One-Way NITF preset studies are available with the Nonisothermal Flow and Conjugate Heat Transfer interfaces. They solve for the fluid flow and heat transfer variables in two separated steps, Stationary for the former preset study, and Time Dependent for the latter.

For a fully coupled approach, use the Stationary, Time Dependent, Stationary with Initialization or Time Dependent with Initialization studies.

See Studies and Solvers in the *COMSOL Multiphysics Reference Manual* for more details.

The Heat Transfer with Surface-to-Surface Radiation Interface

In this section:

- The Heat Transfer with Surface-to-Surface Radiation Multiphysics Interface
- Physics Interface Features
- Coupling Feature

The Heat Transfer with Surface-to-Surface Radiation Multiphysics Interface

When the predefined **Heat Transfer with Surface-to-Surface Radiation** 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 Surface-to-Surface Radiation interfaces to model heat transfer by conduction, convection, and radiation in transparent media.

Stationary, time-domain, and frequency-domain modeling are supported in all space dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Solids** and **Surface-to-Surface Radiation** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling feature.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation in optically thick media. A **Solid** model is active by default on all domains. All functionality to include other heat transfer models, like **Fluid**, is also available.

The Surface-to-Surface Radiation interface provides features to account for 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).

The **Heat Transfer with Surface-to-Surface Radiation** 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 any model
- The Surface-to-Surface Radiation interface

SETTINGS FOR THE 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 feature.

TABLE 8-2: MODIFIED SETTINGS FOR THE HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION
INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Surface-to-Surface Radiation	In the **Model Input** section of the **Diffuse Surface** default feature, the **Temperature**, T, is automatically set to the variable from the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling feature.
Heat Transfer in Solids	In 2D and 1D axisymmetric components, in the **Physical Model** section of the interface, the **Thickness**, d_z, is automatically set to the variable from the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling feature. In 1D components, in the **Physical Model** section of the interface, the **Cross sectional area**, A_c and the **Cross sectional perimeter**, P_c, are automatically set to the variables from the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling feature.
Heat Transfer with Surface-to-Surface Radiation	The **Heat Transfer** and **Surface-to-Surface Radiation** interfaces are preselected

> Note that these settings may be overridden if another predefined coupling is added.

Physics Interface Features

HEAT TRANSFER IN SOLIDS

The available physics features for The Heat Transfer in Solids Interface are listed in Feature Nodes for the Heat Transfer Interface.

SURFACE-TO-SURFACE RADIATION

The available physics features for The Surface-to-Surface Radiation Interface are listed in Feature Nodes for the Surface-to-Surface Radiation Interface.

Coupling Feature

See Heat Transfer with Surface-to-Surface Radiation for details about the multiphysics coupling feature.

The Heat Transfer with Radiation in Participating Media Interface

In this section:

- The Heat Transfer with Radiation in Participating Media Multiphysics Interface
- Physics Interface Features
- Coupling Feature

The Heat Transfer with Radiation in Participating Media Multiphysics Interface

When the predefined **Heat Transfer with Radiation in Participating Media** 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 Radiation in Participating Media interfaces to model heat transfer by conduction, convection, and radiation, including in participating (semitransparent) media.

Stationary, time-domain, and frequency-domain modeling are supported in all space dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Solids** and **Radiation in Participating Media** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the **Heat Transfer with Radiation in Participating Media** multiphysics coupling feature.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation in optically thick media. A **Solid** model is active by default on all domains. All functionality to include other heat transfer models, like **Fluid**, is also available.

The Radiation in participating media interface provides features to account for absorbed, emitted, and scattered radiation in a semitransparent medium. The radiative intensity equations defined in participating media domains are approximated with either the Discrete Ordinates Method or the P1 Approximation. It also computes the heating due to the propagation of rays. A **Participating Medium** model is active by default on all domains, and an **Opaque Surface** node is active on all boundaries. In

addition, when **Discrete ordinates method** is selected, **Continuity on Interior Boundary** is also added.

AS AN ADD-ON MULTIPHYSICS COUPLING

The **Heat Transfer with Radiation in Participating Media** 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 any model
- The Radiation in participating Media interface, with **Participating Medium** model

SETTINGS FOR THE 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 feature.

TABLE 8-3: MODIFIED SETTINGS FOR THE HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Radiation in Participating Media	In the **Model Input** section of the **Participating Medium** default feature, the **Temperature**, T, is automatically set to the variable from the **Heat Transfer with Radiation in Participating Media** multiphysics coupling feature.
Heat Transfer with Radiation in Participating Media	The **Heat Transfer** and **Radiation in Participating Media** interfaces are preselected

Note that these settings may be overridden if another predefined coupling is added.

Physics Interface Features

HEAT TRANSFER IN SOLIDS

The available physics features for The Heat Transfer in Solids Interface are listed in Feature Nodes for the Heat Transfer Interface.

RADIATION IN PARTICIPATING MEDIA

The available physics features for The Radiation in Participating Media Interface are listed in Feature Nodes for the Radiation in Participating Media Interface.

Coupling Feature

See Heat Transfer with Radiation in Participating Media for details about the multiphysics coupling feature.

The Heat Transfer with Radiation in Absorbing-Scattering Media Interface

In this section:

- The Heat Transfer with Radiation in Absorbing-Scattering Media Multiphysics Interface
- Physics Interface Features
- Coupling Feature

The Heat Transfer with Radiation in Absorbing-Scattering Media Multiphysics Interface

When the predefined **Heat Transfer with Radiation in Absorbing-Scattering Media** 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 Radiation in Absorbing-Scattering Media interfaces to model heat transfer by conduction, convection, and radiation, including in participating (semitransparent) media considered to be non-emitting.

Stationary, time-domain, and frequency-domain modeling are supported in all space dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Solids** and **Radiation in Absorbing-Scattering Media** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling feature.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation in optically thick media. A **Solid** model is active by default on all domains. All functionality to include other heat transfer models, like **Fluid**, is also available.

The Radiation in Absorbing-Scattering Media interface provides features to account for absorbed and scattered radiation in a semitransparent medium. Radiative emission is not considered. The radiative intensity equations defined in participating media domains are approximated with either the Discrete Ordinates Method or the P1

Approximation. It also computes the heating due to the propagation of rays. An **Absorbing-Scattering Medium** model is active by default on all domains, and an **Opaque Surface** node is active on all boundaries. In addition, when **Discrete ordinates method** is selected, **Continuity on Interior Boundary** is also added.

AS AN ADD-ON MULTIPHYSICS COUPLING

The **Heat Transfer with Radiation in Absorbing-Scattering Media** 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 any model
- The Radiation in Absorbing-Scattering Media interface, with an **Absorbing-Scattering Medium** model

SETTINGS FOR THE 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 feature.

TABLE 8-4: MODIFIED SETTINGS FOR THE HEAT TRANSFER WITH RADIATION IN ABSORBING-SCATTERING MEDIA INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Radiation in Absorbing-Scattering Media	In the **Model Input** section of the **Absorbing-Scattering Medium** default feature, the **Temperature**, T, is automatically set to the variable from the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling feature.
Heat Transfer with Radiation in Absorbing-Scattering Media	The **Heat Transfer** and **Radiation in Absorbing-Scattering Media** interfaces are preselected.

> Note that these settings may be overridden if another predefined coupling is added.

Physics Interface Features

HEAT TRANSFER IN SOLIDS

The available physics features for The Heat Transfer in Solids Interface are listed in Feature Nodes for the Heat Transfer Interface.

RADIATION IN ABSORBING-SCATTERING MEDIA

The available physics features for The Radiation in Absorbing-Scattering Media Interface are listed in Feature Nodes for the Radiation in Absorbing-Scattering Media Interface.

Coupling Feature

See Heat Transfer with Radiation in Absorbing-Scattering Media for details about the multiphysics coupling feature.

The Heat Transfer with Radiative Beam in Absorbing Media Interface

In this section:

- The Heat Transfer with Radiative Beam in Absorbing Media Multiphysics Interface
- Physics Interface Features
- Coupling Feature

The Heat Transfer with Radiative Beam in Absorbing Media Multiphysics Interface

When the predefined **Heat Transfer with Radiative Beam in Absorbing Media** 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 Radiative Beam in Absorbing Media interfaces to model heat transfer by conduction, convection, and radiation in absorbing (semitransparent) media.

Stationary, time-domain, and frequency-domain modeling are supported in all space dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Solids** and **Radiative Beam in Absorbing Media** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the **Heat Transfer with Radiative Beam in Absorbing Media** multiphysics coupling feature.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation in optically thick media. A **Solid** model is active by default on all domains. All functionality to include other heat transfer models, like **Fluid**, is also available.

The Radiative Beam in Absorbing Media interface provides features to model the attenuation of an incident light within a semitransparent material due to absorption. The beam intensity is determined using the Beer-Lambert Law. As the beam is absorbed it deposits energy which acts as a heat source. An **Absorbing Medium** model is active by default on all domains, and a **Transparent Surface** node is active on all boundaries.

AS AN ADD-ON MULTIPHYSICS COUPLING

The **Heat Transfer with Radiative Beam in Absorbing Media** 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 any model
- The Radiative Beam in absorbing Media interface, with an **Absorbing Medium** model

SETTINGS FOR THE 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 feature.

TABLE 8-5: MODIFIED SETTINGS FOR THE HEAT TRANSFER WITH RADIATION IN ABSORBING MEDIA INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Radiative Beam in Absorbing Media	In the **Model Input** section of the **Absorbing medium** default feature, the **Temperature**, T, is automatically set to the variable from the **Heat Transfer with Radiative Beam in Absorbing Media** multiphysics coupling feature.
Heat Transfer with Radiative Beam in Absorbing Media	The **Heat Transfer** and **Radiative Beam in Absorbing Media** interfaces are preselected

> 🔍 Note that these settings may be overridden if another predefined coupling is added.

Physics Interface Features

HEAT TRANSFER IN SOLIDS

The available physics features for The Heat Transfer in Solids Interface are listed in Feature Nodes for the Heat Transfer Interface.

RADIATIVE BEAM IN ABSORBING MEDIA

The available physics features for The Radiative Beam in Absorbing Media Interface are listed in Settings for the Radiative Beam in Absorbing Media Interface.

Coupling Feature

See Heat Transfer with Radiative Beam in Absorbing Media for details about the multiphysics coupling feature.

The Thermoelectric Effect Interface

In this section:

About 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 domains and on boundaries. In addition, the **Electromagnetic Heating** and the **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, time-domain, and frequency-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, the **Electric Currents** and **Heat Transfer in Solids** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which includes the **Thermoelectric Effect** and **Electromagnetic Heating** multiphysics coupling features. Additional features describing the thermal and electrical effects on boundaries need to be active in each interface to account for the effect also on boundaries.

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. An **Electric Shielding** feature may be added to compute the thermoelectric effect also on boundaries.

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. The thermoelectric effect in domains is added through a contribution to the conductive flux. A **Thin Layer** feature may be added to account for the thermoelectric effect also on boundaries.

AS AN ADD-ON MULTIPHYSICS COUPLING

The Thermoelectric Effect multiphysics coupling is also available when there is any version of the Heat Transfer or Heat Transfer in Shells interface and one of the compatible AC/DC interface:

- 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 Multiphysics adds an empty **Multiphysics** node. You can choose from the following available coupling features: **Thermoelectric Effect** and **Electromagnetic Heating**, 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 8-6: 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 **Boundary Selection** is the same as the exterior and interior boundaries of the **Domain Selection** of the participating physics interfaces. But both a thermal and an electrical model need to be active on the boundaries for the coupling to be applicable. Add for example a **Thin Layer** node under the Heat Transfer interface and an **Electric Shielding** node under the Electric Currents interface to make it applicable.
	The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Coupled Interfaces section of the Thermoelectric Effect node's Settings window.
Electromagnetic Heating	The **Domain Selection** is the same as that of the participating physics interfaces.
	The **Boundary Selection** contains all the boundaries of the participating interfaces.
	The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Coupled Interfaces section of the Thermoelectric Effect node's Settings window.

Coupling Features

See Thermoelectric Effect and Electromagnetic Heating 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 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 Heat and Moisture Transport Interfaces

In this section:

- The Heat and Moisture Transport Multiphysics Interfaces
- The Moist Air Version of the Heat and Moisture Transport Multiphysics Interface
- The Moist Porous Media Version of the Heat and Moisture Transport Multiphysics Interface
- The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface
- Physics Interface Features
- Coupling Feature

The Heat and Moisture Transport Multiphysics Interfaces

The versions of the Heat and Moisture Transport multiphysics interface are:

- The Moist Air Version of the Heat and Moisture Transport Multiphysics Interface
- The Moist Porous Media Version of the Heat and Moisture Transport Multiphysics Interface
- The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface

LATENT HEAT SOURCES

The latent heat sources are obtained by multiplying the evaporation fluxes by the latent heat of evaporation L_v, and are automatically added to the heat transfer equation:

- On boundaries, where the **Wet Surface** or **Moist Surface** features are applied, when the **Include latent heat source on surfaces** check box is selected.
- In domains, where the **Hygroscopic Porous Medium** is active.

DIFFUSIVE FLUX OF THERMAL ENTHALPY

The diffusive flux of thermal enthalpy related to the rate of change of vapor in air is automatically handled in domains where the **Moist Air** or **Hygroscopic Porous Medium** features are applied.

*The Moist Air Version of the Heat and Moisture Transport
Multiphysics Interface*

When the **Moist Air** version of 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 Moist Air and the Moisture
Transport in Air interfaces to model coupled heat and moisture transport in moist air,
by modeling moisture transport by vapor diffusion and convection, and heat transfer
by conduction and convection with moisture-dependent thermodynamics properties.

Stationary, time-domain, and frequency-domain modeling are supported in all space
dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Moist Air** and **Moisture Transport
in Air** 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 Moist Air interface provides features for modeling heat transfer
by conduction, convection, and radiation. The **Moist Air** model, active by default in all
domains, provides in addition the functionality for moisture content dependency of
thermodynamics properties.

The Moisture Transport in Air interface provides features for modeling moisture
transfer by vapor convection and diffusion. A **Moist Air** 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 **Moist Air** model
- Any version of the Moisture Transport interface, with **Moist Air** model

When physics interfaces are added using the predefined couplings, specific settings are included with the physics interfaces and the coupling feature.

TABLE 8-7: MODIFIED SETTINGS FOR THE MOIST AIR VERSION OF THE HEAT AND MOISTURE TRANSPORT INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Moist Air	In the **Thermodynamics, Moist Air** section of the **Moist Air** default feature, the **Input quantity** is set to **Relative humidity**, and the **Relative humidity**, ϕ_w, the **Relative humidity, temperature condition**, T_{ϕ_w}, and the **Relative humidity, absolute pressure condition**, p_{ϕ_w}, are automatically set to the variables from the **Heat and Moisture** multiphysics coupling feature.
Moisture Transport in Air	In the **Model Input** section of the **Moist Air** default feature, the **Temperature**, T, is automatically set to the variable from the **Heat and Moisture** multiphysics coupling feature.
Heat and Moisture	The **Heat Transfer** and **Moisture Transport** interfaces are preselected

Note that these settings may be overridden if another predefined coupling is added.

The Moist Porous Media Version of the Heat and Moisture Transport Multiphysics Interface

When the **Moist Porous Media** version of 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 Moist Porous Media and the Moisture Transport in Porous Media interfaces to model coupled heat and moisture transport in porous media filled with moist air and liquid water, by taking into account heat and moisture storage, latent heat effects, and liquid and gas transport of moisture.

Stationary, time-domain, and frequency-domain modeling are supported in all space dimensions.

As a predefined multiphysics coupling, the **Heat Transfer in Moist Porous Media** and **Moisture Transport in Porous Media** 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 Moist Porous Media interface provides features for modeling heat transfer by conduction, convection, and radiation. The **Moist Porous Medium** 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 in Porous Media interface provides features for modeling moisture transfer by liquid transport (capillary and convective flow), and vapor convection and diffusion. An **Hygroscopic 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 **Moist Porous Medium** model

• Any version of the Moisture Transport interface, with **Hygroscopic Porous Medium** model

When physics interfaces are added using the predefined couplings, specific settings are included with the physics interfaces and the coupling feature.

TABLE 8-8: MODIFIED SETTINGS FOR THE MOIST POROUS MEDIA VERSION OF THE HEAT AND MOISTURE TRANSPORT INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Moist Porous Media	In the **Thermodynamics, Moist Air** section of the **Moist Air** default subfeature of **Moist Porous Medium**, the **Input quantity** is set to **Relative humidity**, and the **Relative humidity**, ϕ_w, the **Relative humidity, temperature condition**, T_{ϕ_w}, and the **Relative humidity, absolute pressure condition**, p_{ϕ_w}, are automatically set to the variables from the **Heat and Moisture** multiphysics coupling feature.
	In the **Liquid Water Properties** section of the **Liquid Water** default subfeature of **Moist Porous Medium**, the **Liquid water saturation**, s_l, and the **Velocity field, liquid water**, $\mathbf{u}_l$, are automatically set to the variables from the **Heat and Moisture** multiphysics coupling feature.
Moisture Transport in Porous Media	In the **Model Input** section of the **Hygroscopic Porous Medium** default feature, the **Temperature**, T, is automatically set to the variable from the **Heat and Moisture** multiphysics coupling feature.
Heat and Moisture	The **Heat Transfer** and **Moisture Transport** interfaces are preselected

> Note that these settings may be overridden if another predefined
> coupling is added.

The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface

When the **Building Materials** version of 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 in Building Materials 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 diffusive transport of moisture.

Stationary, time-domain, and frequency-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 in Building Materials** 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 in Building Materials interface provides features for modeling moisture transfer by liquid transport (capillary flow) and vapor diffusion. A **Building Material** 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
- Any version of the Moisture Transport interface, with **Building Material** model

---

📄   See Building Material Library in the *COMSOL Multiphysics Reference Manual* for a description of materials containing the thermal and hygroscopic properties to be used with the Building Materials version of the Heat and Moisture Transport multiphysics interface.

When physics interfaces are added using the predefined couplings, specific settings are included with the physics interfaces and the coupling feature.

TABLE 8-9: MODIFIED SETTINGS FOR THE BUILDING MATERIALS VERSION OF THE HEAT AND MOISTURE TRANSPORT INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Building Materials	In the **Model Input** section of the **Building Material** default feature, the **Relative humidity**, ϕ_w, is automatically set to the variable from the **Heat and Moisture** multiphysics coupling feature.
Moisture Transport in Building Materials	In the **Model Input** section of the **Building Material** default feature, the **Temperature**, T, is automatically set to the variable from the **Heat and Moisture** multiphysics coupling feature.
Heat and Moisture	The **Heat Transfer** and **Moisture Transport** interfaces are preselected

> Note that these settings may be overridden if another predefined coupling is added.

Physics Interface Features

HEAT TRANSFER

The available physics features for the Heat Transfer interface are listed in Feature Nodes for the Heat Transfer Interface.

MOISTURE TRANSPORT

The available physics features for the Moisture Transport interface are listed in Feature Nodes for the Moisture Transport Interface.

Coupling Feature

See Heat and Moisture for details about the multiphysics coupling feature.

The Moisture Flow Interfaces

In this section:

The Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces

These versions of the Moisture Flow interface model moisture transport in air by vapor diffusion and convection.

When a Moisture Flow (≋) multiphysics interface is added from the **Chemical Species Transport>Moisture Flow** branch of the **Model Wizard** or **Add Physics** windows, one of the **Single-Phase Flow** interfaces (laminar or turbulent flow) and a Moisture Transport in air interface are added to the Model Builder.

In addition, the **Multiphysics** node is added, which includes the **Moisture Flow** multiphysics coupling feature.

> The Multiphysics Branch in the *COMSOL Multiphysics Reference Manual*.

- The Laminar Flow interface (≋) combines a Moisture Transport in Air interface with a Laminar Flow interface.
- The Turbulent Flow, Algebraic yPlus interface (≋) combines a Moisture Transport in Air interface with a Turbulent Flow, Algebraic yPlus interface.
- The Turbulent Flow, L-VEL interface (≋) combines a Moisture Transport in Air interface with a Turbulent Flow, L-VEL interface.
- The Turbulent Flow, k-ε interface (≋) combines a Moisture Transport in Air interface with a Turbulent Flow, k-ε interface.

- The Turbulent Flow, k-ω interface (⬢) combines a Moisture Transport in Air interface with a Turbulent Flow, k-ω interface.

- The Turbulent Flow, Low Re k-ε interface (⬢) combines a Moisture Transport in Air interface with a Turbulent Flow, Low Re k-ε interface.

- The Turbulent Flow, SST interface (⬢) combines a Moisture Transport in Air interface with a Turbulent Flow, SST interface.

- The Turbulent Flow, Spalart-Allmaras interface (⬢) combines a Moisture Transport in Air interface with a Turbulent Flow, Spalart-Allmaras interface.

Stationary and time-domain modeling are supported in all space dimensions.

SETTINGS FOR THE 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 feature.

TABLE 8-10: MODIFIED SETTINGS FOR THE MOISTURE FLOW INTERFACES, LAMINAR AND TURBULENT FLOW VERSIONS

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Moisture Transport in Air	In the **Model Input** section of the **Moist Air** default domain feature, the **Absolute pressure**, p_A, and the **Velocity field**, **u**, are automatically set to the variables from the **Moisture Flow** multiphysics coupling feature.
Laminar Flow / Turbulent Flow	In the **Fluid Properties** default domain feature, the **Density**, ρ, and the **Dynamic viscosity**, μ, are automatically set to the moist air variables from the **Moisture Flow** multiphysics coupling feature.
Moisture Flow	The **Fluid flow** and **Moisture Transport** interfaces are preselected

> 🔍 Note that these settings may be overridden if another predefined coupling is added.

ON THE CONSTITUENT PHYSICS INTERFACES

The Moisture Transport in Air interface provides features for modeling moisture transfer by vapor convection and diffusion. A **Moist Air** model is active by default on all domains.

The Laminar Flow interface solves the Navier-Stokes equations for conservation of momentum and the continuity equation for conservation of mass. A **Fluid** model is active by default on all domains.

The different versions of the Turbulent Flow interface solve the Reynolds averaged Navier-Stokes equations for conservation of momentum and the continuity equation for conservation of mass. A **Fluid Properties** model is active by default on all domains. Turbulence effects are modeled in different ways:

- The Turbulent Flow, Algebraic yPlus interface uses an enhanced viscosity model based on the local wall distance. The physics interface therefore includes a wall distance equation.

- The Turbulent Flow, L-VEL interface uses an enhanced viscosity model based on the local wall distance. The physics interface therefore includes a wall distance equation.

- The Turbulent Flow, k-ε interface uses the standard two-equation k-ε model with realizability constraints. Flow close to walls is modeled using wall functions.

- The Turbulent Flow, k-ω interface uses the Wilcox revised two-equation k-ω model with realizability constraints. Flow close to walls is modeled using wall functions.

- The Turbulent Flow, Low Re k-ε interface uses the AKN two-equation k-ε model with realizability constraints. The AKN model is a so-called low-Reynolds number model, which means that it resolves the flow all the way down to the wall. The AKN model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

AS AN ADD-ON MULTIPHYSICS COUPLING

The **Moisture Flow** multiphysics coupling is also available when there is at least one of each of the following interfaces with the specified model activated:

- A Laminar Flow or any version of the Turbulent Flow interface, with **Fluid** model
- Any version of the Moisture Transport interface, with **Moist Air** model

Physics Interface Features for the Laminar and Turbulent Flow Versions

MOISTURE TRANSPORT IN AIR

The available physics features for The Moisture Transport in Air Interface are listed in Feature Nodes for the Moisture Transport Interface in the *Heat Transfer Module User's Guide*.

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-ε 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-ω

The available physics features for The Turbulent Flow, k-ω 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-ε Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

The Moisture Flow, Brinkman Equations Multiphysics Interface

This version of the Moisture Flow interface models moisture transport in the gas phase of a moist porous medium, by vapor diffusion and convection. When a Moisture Flow, Brinkman Equations ( ) multiphysics interface is added from the **Chemical Species Transport>Moisture Flow** branch of the **Model Wizard** or **Add Physics** windows, a Brinkman Equations interface and a Moisture Transport in Porous Media interface are added to the Model Builder.

In addition, the **Multiphysics** node is added, which includes the **Moisture Flow** multiphysics coupling feature.

Q | The Multiphysics Branch in the *COMSOL Multiphysics Reference Manual*.

Stationary and time-domain modeling are supported in all space dimensions.

SETTINGS FOR THE 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 feature.

TABLE 8-11: MODIFIED SETTINGS FOR THE MOISTURE FLOW, BRINKMAN EQUATIONS INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Moisture Transport in Porous Media	In the **Hygroscopic Porous Medium** default domain feature, the **Absolute pressure**, p_A (**Model Input** section) and the **Velocity field**, $\mathbf{u}_g$ (**Moist Air Properties** section) are automatically set to the variables from the **Moisture Flow** multiphysics coupling feature.
Brinkman Equations	In the **Fluid and Matrix Properties** default domain feature, the **Density**, ρ, and the **Dynamic viscosity**, μ, are automatically set to the moist air variables from the **Moisture Flow** multiphysics coupling feature.
Moisture Flow	The **Fluid flow** and **Moisture Transport** interfaces are preselected

---

Q | Note that these settings may be overridden if another predefined coupling is added.

ON THE CONSTITUENT PHYSICS INTERFACES

The Moisture Transport in Porous Media interface provides features for modeling moisture transport in media filled with moist air and liquid water, with a time-dependent liquid saturation. Transport is done by vapor convection and diffusion in the gas phase, and convection and capillarity in the liquid water phase. An **Hygroscopic Porous Medium** model is active by default on all domains.

The Brinkman Equations interface computes the fluid velocity and pressure fields of a single-phase flow in porous media in the laminar flow regime. A **Fluid and Matrix Properties** model is active by default on all domains.

Physics Interface Features for the Brinkman Equations Version

MOISTURE TRANSPORT IN POROUS MEDIA

The available physics features for The Moisture Transport in Porous Media Interface are listed in Feature Nodes for the Moisture Transport Interface in the *Heat Transfer Module User's Guide*.

BRINKMAN EQUATIONS

The available physics features for The Brinkman Equations Interface are listed in the section Domain, Boundary, Point, and Pair Nodes for the Brinkman Equations Interface in the *CFD Module User's Guide*.

Preset Studies

The Stationary, One-Way MF and Time Dependent, One-Way MF preset studies are available with the Moisture Flow interfaces. They solve for the fluid flow and moisture transport variables in two separated steps, Stationary for the former preset study, and Time Dependent for the latter.

For a fully-coupled approach, use the Stationary, Time Dependent, Stationary with Initialization, or Time Dependent with Initialization studies.

See Studies and Solvers in the *COMSOL Multiphysics Reference Manual* for more details.

Coupling Feature

See Moisture Flow for details about the multiphysics coupling feature.

The Heat and Moisture Flow Interfaces

In this section:

- The Heat and Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces
- Physics Interface Features for the Laminar and Turbulent Flow Versions
- The Heat and Moisture Flow, Brinkman Equations Interface
- Physics Interface Features for the Brinkman Equations Version
- Coupling Features

The Heat and Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces

The Heat and Moisture Flow interfaces model heat and moisture transport in air by laminar or turbulent flows.

When a Heat and Moisture Flow ( ) multiphysics interface is added from the **Heat Transfer>Heat and Moisture Transport>Heat and Moisture Flow** branch of the **Model Wizard** or **Add Physics** windows, one of the **Single-Phase Flow** interfaces (laminar or turbulent flow), a Heat Transfer in Moist Air interface, and a Moisture Transport in air interface are added to the Model Builder.

In addition, the **Multiphysics** node is added, which includes the **Moisture Flow**, **Heat and Moisture**, and **Nonisothermal Flow** multiphysics coupling features.

> The Multiphysics Branch in the *COMSOL Multiphysics Reference Manual*.

- The Laminar Flow interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Laminar Flow interface.
- The Turbulent Flow, Algebraic yPlus interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, Algebraic yPlus interface.

- The Turbulent Flow, L-VEL interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, L-VEL interface.

- The Turbulent Flow, k-ε interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, k-ε interface.

- The Turbulent Flow, Realizable k-ε interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, Realizable k-ε interface.

- The Turbulent Flow, k-ω interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, k-ω interface.

- The Turbulent Flow, SST interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, SST interface.

- The Turbulent Flow, Low Re k-ε interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, Low Re k-ε interface.

- The Turbulent Flow, Spalart-Allmaras interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, Spalart-Allmaras interface.

- The Turbulent Flow, v2-f interface ( ) combines a Heat Transfer in Moist Air interface, a Moisture Transport in Air interface, and a Turbulent Flow, v2-f interface.

Stationary and time-domain modeling are supported in all space dimensions.

SETTINGS FOR THE 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 feature.

TABLE 8-12: MODIFIED SETTINGS FOR THE HEAT AND MOISTURE FLOW INTERFACES, LAMINAR AND TURBULENT FLOW VERSIONS

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Moist Air	In the **Model Input** section of the **Moist Air** default domain feature, the **Absolute pressure**, p_A, and the **Velocity field**, **u**, are automatically set to the variables from the **Nonisothermal Flow** multiphysics coupling feature. The Concentration, c, is automatically set to the variable from the **Heat and Moisture** multiphysics coupling feature.
	In the **Thermodynamics, Moist Air** section of the **Moist Air** default feature, the **Input quantity** is set to **Relative humidity**. The **Relative humidity**, ϕ_w, the **Relative humidity, temperature condition**, T_{ϕ_w}, and the **Relative humidity, absolute pressure condition**, p_{ϕ_w}, are automatically set to the variables from the **Heat and Moisture** multiphysics coupling feature.
	The latent heat sources are automatically handled on boundaries where **Wet Surface** or **Moist Surface** features are applied.
Moisture Transport in Air	In the **Model Input** section of the **Moist Air** default domain feature, the **Absolute pressure**, p_A, and the **Velocity field**, **u**, are automatically set to the variables from the **Moisture Flow** multiphysics coupling feature. The Temperature, T, is automatically set to the variable from the **Heat and Moisture** multiphysics coupling feature.
Laminar Flow / Turbulent Flow	In the **Fluid Properties** default domain feature, the **Density**, ρ, and the **Dynamic viscosity**, μ, are automatically set to the variables from the **Moisture Flow** multiphysics coupling feature.
Moisture Flow	The **Fluid flow** and **Moisture Transport** interfaces are preselected.
Heat and Moisture	The **Heat Transfer** and **Moisture Transport** interfaces are preselected.
Nonisothermal Flow	The **Fluid flow** and **Heat Transfer** interfaces are preselected.

	Note that these settings may be overridden if another predefined coupling is added.

ON THE CONSTITUENT PHYSICS INTERFACES

The Heat Transfer in Moist Air interface provides features for modeling heat transfer by conduction, convection, and radiation. A **Moist Air** model is active by default on all domains.

The Moisture Transport in Air interface provides features for modeling moisture transport by vapor convection and diffusion. A **Moist Air** model is active by default on all domains.

The Laminar Flow interface solves the Navier-Stokes equations for conservation of momentum and the continuity equation for conservation of mass. A **Fluid Properties** model is active by default on all domains.

The different versions of the Turbulent Flow interface solve the Reynolds averaged Navier-Stokes equations for conservation of momentum, the heat transfer equation, and the continuity equation for conservation of mass. A **Fluid Properties** model is active by default on all domains. Turbulence effects are modeled in different ways:

- The Turbulent Flow, Algebraic yPlus interface uses an enhanced viscosity model based on the local wall distance. The physics interface therefore includes a wall distance equation.

- The Turbulent Flow, L-VEL interface uses an enhanced viscosity model based on the local wall distance. The physics interface therefore includes a wall distance equation.

- The Turbulent Flow, k-ε interface uses the standard two-equation k-ε model with realizability constraints. Flow close to walls is modeled using wall functions.

- The Turbulent Flow, k-ω interface uses the Wilcox revised two-equation k-ω model with realizability constraints. Flow close to walls is modeled using wall functions.

- The Turbulent Flow, Low Re k-ε interface uses the AKN two-equation k-ε model with realizability constraints. The AKN model is a so-called low-Reynolds number model, which means that it resolves the flow all the way down to the wall. The AKN model depends on the distance to the closest wall. The physics interface therefore includes a wall distance equation.

Physics Interface Features for the Laminar and Turbulent Flow Versions

HEAT TRANSFER IN MOIST AIR

The available physics features for The Heat Transfer in Moist Air Interface are listed in Feature Nodes for the Heat Transfer Interface. Also see Moist Air (Heat Transfer Interface) for details about that feature.

MOISTURE TRANSPORT IN AIR

The available physics features for The Moisture Transport in Air Interface are listed in Settings for the Moisture Transport Interface. Also see Moist Air (Moisture Transport Interface) 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-ε 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-ω

The available physics features for The Turbulent Flow, k-ω Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

The available physics features for The Turbulent Flow, Low Re k-$\mathcal{E}$ Interface are listed in the section Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow in the *CFD Module User's Guide*.

The Heat and Moisture Flow, Brinkman Equations Interface

The Heat and Moisture Flow, Brinkman Equations interface models heat and moisture transport in porous media filled with liquid water air moist air.

When a Heat and Moisture Flow, Brinkman Equations ( ) multiphysics interface is added from the **Heat Transfer>Heat and Moisture Transport>Heat and Moisture Flow** branch of the **Model Wizard** or **Add Physics** windows, a Brinkman Equations interface, a Heat Transfer in Moist Porous Media interface, and a Moisture Transport in Porous Media interface are added to the Model Builder.

In addition, the **Multiphysics** node is added, which includes the **Moisture Flow**, **Heat and Moisture**, and **Nonisothermal Flow** multiphysics coupling features.

⊘	The Multiphysics Branch in the *COMSOL Multiphysics Reference Manual*.

Stationary and time-domain modeling are supported in all space dimensions.

When physics interfaces are added using the predefined couplings, specific settings are included with the physics interfaces and the coupling feature.

TABLE 8-13: MODIFIED SETTINGS FOR THE HEAT AND MOISTURE FLOW, BRINKMAN EQUATIONS INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Moist Porous Media	In the **Moist Air** default subfeature of **Moist Porous Medium**, the **Absolute pressure**, p_A (**Model Input** section) and the **Velocity field, $\mathbf{u}_g$** (**Heat Convection** section) are automatically set to the variables from the **Nonisothermal Flow** multiphysics coupling feature. In the **Thermodynamics, Moist Air** section, the **Input quantity** is set to **Relative humidity**, and the **Relative humidity**, ϕ_w, the **Relative humidity, temperature condition**, T_{ϕ_w}, and the **Relative humidity, absolute pressure condition**, p_{ϕ_w}, are automatically set to the variables from the **Heat and Moisture** multiphysics coupling feature.
	In the **Liquid Water Properties** section of the **Liquid Water** default subfeature of **Moist Porous Medium**, the **Liquid water saturation**, s_l, and the **Velocity field, liquid water, $\mathbf{u}_l$**, are automatically set to the variables from the **Heat and Moisture** multiphysics coupling feature.
	The latent heat sources are automatically handled on boundaries where **Wet Surface** or **Moist Surface** features are applied.
Moisture Transport in Porous Media	In the **Hygroscopic Porous Medium** default domain feature, the **Absolute pressure**, p_A (**Model Input** section) and the **Velocity field, $\mathbf{u}_g$** (**Moist Air Properties** section) are automatically set to the variables from the **Moisture Flow** multiphysics coupling feature. In the **Model Input** section, the Temperature, T, is automatically set to the variable from the **Heat and Moisture** multiphysics coupling feature.

TABLE 8-13: MODIFIED SETTINGS FOR THE HEAT AND MOISTURE FLOW, BRINKMAN EQUATIONS INTERFACE

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Brinkman Equations	In the **Fluid and Matrix Properties** default domain feature, the **Temperature**, T, and the **Density**, ρ, are set to the variables from the **Nonisothermal Flow** multiphysics coupling feature, and the **Dynamic viscosity**, μ, is set to the variable from the **Moisture Flow** multiphysics coupling feature. Note that the density and viscosity are those of the moist air phase.
Moisture Flow	The **Fluid flow** and **Moisture Transport** interfaces are preselected.
Heat and Moisture	The **Heat Transfer** and **Moisture Transport** interfaces are preselected.
Nonisothermal Flow	The **Fluid flow** and **Heat Transfer** interfaces are preselected.

> 🔍 Note that these settings may be overridden if another predefined coupling is added.

ON THE CONSTITUENT PHYSICS INTERFACES

The Heat Transfer in Moist Porous Media interface provides features for modeling heat transfer by conduction, convection, and radiation. The **Moist Porous Medium** 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 in Porous Media interface provides features for modeling moisture transfer by liquid transport (capillary and convective flow), and vapor convection and diffusion. An **Hygroscopic Porous Medium** model is active by default on all domains.

The Brinkman Equations interface computes the fluid velocity and pressure fields of a single-phase flow in porous media in the laminar flow regime. A **Fluid and Matrix Properties** model is active by default on all domains.

Physics Interface Features for the Brinkman Equations Version

HEAT TRANSFER IN MOIST POROUS MEDIA

The available physics features for The Heat Transfer in Moist Porous Media Interface are listed in Feature Nodes for the Heat Transfer Interface. Also see Moist Porous Medium for details about that feature.

MOISTURE TRANSPORT IN POROUS MEDIA

The available physics features for The Moisture Transport in Porous Media Interface are listed in Feature Nodes for the Moisture Transport Interface in the *Heat Transfer Module User's Guide*.

BRINKMAN EQUATIONS

The available physics features for The Brinkman Equations Interface are listed in the section Domain, Boundary, Point, and Pair Nodes for the Brinkman Equations Interface in the *CFD Module User's Guide*.

Coupling Features

See Moisture Flow, Heat and Moisture, and Nonisothermal Flow for details about the multiphysics coupling features.

The Joule Heating Interface

In this section:

- The Joule Heating Interface
- Coupling Feature

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 Feature

See Electromagnetic Heating for a description of the multiphysics coupling.

The Laser Heating Interface

In this section:

- The Laser Heating Interface
- Coupling Feature

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 Feature

See Electromagnetic Heating for a description of the multiphysics coupling.

The Induction Heating Interface

In this section:

- The Induction Heating Interface
- Coupling Feature

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 Feature

See Electromagnetic Heating for a description of the multiphysics coupling.

The Microwave Heating Interface

In this section:

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 Feature

See Electromagnetic Heating for a description of the multiphysics coupling.

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 Branch 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 Heating
- Heat and Moisture
- Heat Transfer with Radiation in Participating Media
- Heat Transfer with Radiation in Absorbing-Scattering Media
- Heat Transfer with Radiative Beam in Absorbing Media

- Moisture Flow
- Nonisothermal Flow
- Thermal Expansion
- Thermoelectric Effect

For a detailed overview of the functionality available in each product, visit
https://www.comsol.com/products/specifications/

Electromagnetic Heating

Use the **Electromagnetic Heating** multiphysics coupling ( ) to account for electromagnetic volumetric and surface losses in the heat equation.

See Electromagnetic Heating in the *COMSOL Multiphysics Reference Manual* for a description of this multiphysics coupling in the context of Joule heating and laser heating modeling.

See Electromagnetic Heating in the *AC/DC Module User's Guide* for a description of this multiphysics coupling in the context of induction heating modeling.

See Electromagnetic Heating 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
- *Contact Switch*: Application Library path **Heat_Transfer_Module/ Thermal_Contact_and_Friction/contact_switch**

Context Menus
Multiphysics>Electromagnetic Heating

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

Heat and Moisture

Use this multiphysics coupling (⬛) to model coupled heat and moisture transfer in the following media:

• In building materials, by taking into account heat and moisture storage, latent heat effects, and transport of heat and moisture. This coupling is available when a **Building Material** feature of the Heat Transfer interface and a **Building Material** feature of the Moisture Transport interface are active on the same domain. It 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.

• In moist air, by modeling moisture transport by vapor diffusion and convection, and heat transfer by conduction and convection. The thermodynamics properties of moist air depend on the moisture content, whereas the temperature is used to define the saturation conditions for vapor concentration. This coupling is available when a

Moist Air feature of the Heat Transfer interface and a **Moist Air** feature of the Moisture Transport interface are active on the same domain.

- In moist porous media, by taking into account heat and moisture storage, latent heat effects, and transport of heat and moisture. This coupling is available when a **Moist Porous Medium** feature of the Heat Transfer interface and an **Hygroscopic Porous Medium** feature of the Moisture Transport interface are active on the same domain. It can be applied to the computation of different moisture variations phenomena in porous media filled with moist air and liquid water.

The **Heat and Moisture** coupling synchronizes the features from the Heat Transfer and Moisture Transport interfaces:

- It defines the relative humidity ϕ_w (with corresponding reference temperature and pressure) in order to set the corresponding inputs in the **Moist Air** 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.

- In building materials domains, 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.

- In moist air domains, it computes the latent heat source due to evaporation and condensation fluxes on surfaces, and adds it in the heat transfer equation.

- In porous media domains, it defines the liquid water saturation s_l and velocity field $\mathbf{u}_l$ in order to set the corresponding inputs in the **Liquid Water** feature of the Heat Transfer interface.

- In porous media domains, it computes the latent heat source due to evaporation and condensation, and adds it in the heat transfer equation, in the **Moist Porous Medium** feature of the Heat Interface.

- In porous media domains, it computes the diffusive flux of thermal enthalpy due to the rate of change of vapor and air in the mixture, which is used in the **Moist Air** subfeture of the **Moist Porous Medium** feature of the Heat Interface.

- In porous media domains, it computes the capillary flux of liquid water, which is used in the **Liquid Water** subfeature of the **Moist Porous Medium** feature of the Heat 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 and 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.

LATENT HEAT

The multiphysics coupling adds the following latent heat source q_{evap} on surfaces due to evaporation and condensation:

$$q_{evap} = L_v g_{evap}$$

where g_{evap} is the evaporative flux and L_v is the latent heat of evaporation.

To neglect the latent heat effects in the heat transfer equation, clear the **Include latent heat source on surfaces** check box.

> *Evaporative Cooling of Water*: Application Library path
> **Heat_Transfer_Module/Phase_Change/evaporative_cooling**

Context Menus
Multiphysics>Heat and Moisture

when a heat transfer interface with **Building Material** feature is active together with a moisture transport interface with **Building Material** feature; or when a heat transfer interface with **Moist Air** feature is active together with a moisture transport interface with **Moist Air** feature; or when a heat transfer interface with **Moist Porous Medium** feature is active together with a moisture transport interface with **Hygroscopic Porous Medium** feature.

Heat Transfer with Radiation in Participating Media

Use the **Heat Transfer with Radiation in Participating Media** multiphysics coupling ( ) to account for heat transfer by conduction, convection, and radiation in semitransparent (participating) media. The radiative intensity equations are approximated either by the Discrete Ordinates Method or the P1 Approximation, depending on the **Radiation discretization method** selected in the **Participating Media Settings** section of the coupled Radiation in Participating Media interface.

It adds the radiative heat source term Q (SI unit: W/m^3) to the heat transfer equation, defined by:

$$Q = \kappa(G - 4\pi I_b(T))$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2).
- $I_b(T)$ 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: $W/(m^2 \cdot K^4)$).
- T is the temperature (SI unit: K)

It also adds the radiative heat flux on boundaries, depending on the selected discretization method. See Opaque Surface (Radiation in Participating Media and

Radiation in Absorbing-Scattering Media Interfaces) for the definition of the radiative heat flux.

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 htrpm1.

COUPLED INTERFACES

This section defines the physics involved in the **Heat Transfer with Radiation in Participating Media** multiphysics coupling.

Select the **Heat transfer** interface associated to the temperature dependent variable. Select the **Radiation in participating media** interface associated to the radiative intensity variable (with Discrete Ordinates Method) or the incident radiation variable (P1 approximation).

> 🔲 In 1D, 1D-axi, and 2D, when the **Heat Transfer with Radiation in Participating Media** multiphysics coupling is active while solving, it disables all the out of plane features (Out-of-Plane Heat Flux, Out-of-Plane Radiation, Thickness, Cross Section) from the heat transfer interface that is selected in the multiphysics coupling. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

LOCATION IN USER INTERFACE

Context Menus

Multiphysics>Heat Transfer with Radiation in Participating Media

when a Heat Transfer interface or a Slip Flow interface with any domain feature is added together with the Radiation in Participating Media interface with **Radiation in Participating Media** feature.

Heat Transfer with Radiation in Absorbing-Scattering Media

Use the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling (⬛) to account for heat transfer by conduction, convection, and radiation in semitransparent (participating) media considered to be non-emitting. The radiative intensity equations are approximated either by the Discrete Ordinates Method or the P1 Approximation, depending on the **Radiation discretization method** selected in the **Participating Media Settings** section of the coupled Radiation in Absorbing-Scattering Media interface.

It adds the radiative heat source term Q (SI unit: W/m^3) to the heat transfer equation, defined by:

$$Q = \kappa G$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2).

It also adds the radiative heat flux on boundaries, depending on the selected discretization method. See Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) for the definition of the radiative heat flux.

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 htrasm1.

COUPLED INTERFACES

This section defines the physics involved in the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling.

Select the **Heat transfer** interface associated to the temperature dependent variable. Select the **Radiation in absorbing-scattering media** interface associated to the radiative

intensity variable (with Discrete Ordinates Method) or the incident radiation variable (P1 approximation).

> [!] In 1D, 1D-axi, and 2D, when the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling is active while solving, it disables all the out of plane features (Out-of-Plane Heat Flux, Out-of-Plane Radiation, Thickness, Cross Section) from the heat transfer interface that is selected in the multiphysics coupling. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

LOCATION IN USER INTERFACE

Context Menus
Multiphysics>Heat Transfer with Radiation in Absorbing-Scattering Media

when a heat transfer interface with any domain feature is added together with the Radiation in Absorbing-Scattering Media interface with **Absorbing-Scattering Medium** feature.

Heat Transfer with Radiative Beam in Absorbing Media

Use the **Heat Transfer with Radiative Beam in Absorbing Media** multiphysics coupling ( ) to model heat transfer by conduction, convection, and radiation in absorbing (semitransparent) media. The attenuation of an incident beam intensity within a semitransparent material due to absorption is determined using the Beer-Lambert Law. As the beam is absorbed it deposits energy which acts as a heat source.

It adds the radiative heat source term Q (SI unit: W/m^3) to the heat transfer equation, defined by:

$$Q = \sum_i \kappa I_i$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- I_i is the ith beam intensity (SI unit: W/m^2), solution of the Beer-Lambert equation.

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 `htrbam1`.

COUPLED INTERFACES

This section defines the physics involved in the **Heat Transfer with Radiative Beam in Absorbing Media** multiphysics coupling.

Select the **Heat transfer in solids** interface associated to the temperature dependent variable. Select the **Radiative Beam in Absorbing Media** interface associated to the radiative beam intensity variable.

LOCATION IN USER INTERFACE

Context Menus

Multiphysics>Heat Transfer with Radiative Beam in Absorbing Media

when a heat transfer interface with any domain feature is added together with the Radiation in Absorbing Media interface with **Absorbing Medium** feature.

Moisture Flow

Use the **Moisture Flow** multiphysics coupling (≈) to simulate fluid flows in free and porous media, where the fluid properties (density, viscosity) depend on moisture content. Models can also include moisture transport in building materials. The physics interface supports low Mach numbers (typically less than 0.3).

The Moisture Flow interface solves for conservation of vapor concentration, mass and momentum in air. It synchronizes the features from the **Moisture Transport** and **Fluid**

Flow interfaces when a turbulent flow regime is defined. It also complements the **Wall**, **Screen** and **Interior Fan** features from the flow interface to account for moisture effects.

When considering an hygroscopic porous medium filled with moist air and liquid water, the coupling between the **Moisture Transport** and **Fluid Flow** interfaces applies on the moist air phase only. This means that the synchronized fluid properties (density, viscosity) are the moist air properties, and that the porosity set in the **Fluid Flow** interface should account for the presence of the liquid water phase in the porous medium. This can be done by multiplying the dry material porosity by (1-mt.sl), where the mt.sl variable is the liquid water saturation defined in 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 mf1.

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 moisture flow, or select **All domains** as needed.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Fluid flow** and **Moisture Transport** 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, **Moisture Transport in Air** is deleted — then the **Moisture Transport** list defaults to **None** as there is nothing to couple to.

MOISTURE TRANSPORT AT WALLS

This section is available for free and porous media flows, when the **Mixture type for moist air** is set to **Concentrated species** in the moisture transport interface.

By selecting the **Account for Stefan velocity at walls** check box, you add a velocity contribution to the fluid flow due to the vapor flux at walls.

When prescribing a vapor flux $\mathbf{g}_w$ at boundaries in the moisture transport interface, using the Moisture Content, Moisture Flux, Wet Surface, or Moist Surface features, the **Moisture Flow** coupling node computes the Stefan velocity u_{Stefan} and applies it in the **Wall** features of the fluid flow interface, at the corresponding boundaries.

The Stefan velocity is defined as

$$\rho_g u_{\text{Stefan}} = \mathbf{n} \cdot \mathbf{g}_w$$

where ρ_g is the moist air density.

At low temperature conditions, the vapor content in moist air stays relatively small, even for high relative humidity, and the Stefan flow may be neglected. By default, the check box is not selected.

MOISTURE TRANSPORT TURBULENCE MODEL

This section is available for free media flow, when the fluid flow interface uses a turbulence model. In this case, the diffusive moisture flux is defined as

$$\mathbf{g}_w = -M_v(D + D_T)\nabla c_v$$

to account for the turbulent mixing caused by the eddy diffusivity in moisture convection, with the turbulent diffusivity defined as

$$D_T = \frac{\nu_T}{Sc_T}$$

where ν_T is defined by the flow interface, and the turbulent Schmidt number Sc_T depends on the **Moisture transport turbulence model**.

Select an option from the **Moisture transport turbulence model** list: **Kays-Crawford** (the default) or **User-defined turbulent Schmidt number**.

For **User-defined turbulent Schmidt number**, enter a **Turbulent Schmidt number** Sc_T (dimensionless). The turbulent kinematic viscosity ν_T is taken directly from the fluid flow interface.

For **Kays-Crawford**, see Kays-Crawford Model for Turbulent Diffusivity for details about the definition of Sc_T.

The **Turbulence model type** used by the fluid flow interface can be displayed by selecting the **Show or Hide Physics Property Settings** button at the right of the **Fluid flow** list.

⚠️ When an interface is selected from the **Moisture Transport** list, some of its model inputs are forced with values from the **Moisture 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 moisture transport or fluid flow interface should be used in at most one **Moisture Flow** node. In cases where multiple fluid flow interfaces are used, an equal number of moisture transport interfaces and **Moisture 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 reestablish the coupling, you need to choose the physics interface again from the **Fluid flow** or **Moisture Transport** lists. This behavior is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Workflow in the *COMSOL Multiphysics Reference Manual*.

	Evaporative Cooling of Water: Application Library path
▥	**Heat_Transfer_Module/Phase_Change/evaporative_cooling**

Nonisothermal Flow

Use the **Nonisothermal Flow** 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 Nonisothermal 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 defines p and $\mathbf{u}$ variables in order to set the **Absolute pressure** in the **Model Input** section and the **Velocity field** in the **Heat Convection** section of the **Fluid** feature and subfeature, in the Heat Transfer interface. In addition, it provides all the fluids quantities that may be needed by the Heat Transfer interface (for example, viscosity, turbulence parameters).

In the Fluid Flow interface, it sets the **Temperature** in the **Model Input** section and defines the **Density** in the **Fluid Properties** section of the **Fluid Properties** and **Fluid and Matrix Properties** features.

It synchronizes also the definition of the reference temperature to be used for incompressible flows, and 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.

In addition, it also accounts for the multiphysics stabilization terms, for the heat transfer changes in the turbulent regime (for example, thermal wall functions), for work due to pressure forces and viscous dissipation, and for natural convection, including a Boussinesq approximation.

> 📋 The **Nonisothermal Flow** coupling node triggers pseudo time stepping when **Use pseudo time stepping for stationary equation form** in the **Fluid Flow** interface is set to **Automatic from physics**.

> 📋 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.
>
> Finally, when one of the physics interfaces or the multiphysics coupling is not solved in a study step, then the stabilization of each solved physics is used instead of the coupled stabilization, and the solver suggestions are uncoupled.

> 📋 The pressure, velocity, and temperature variables of the **Nonisothermal Flow** coupling node are set to the **Common Model Input** values of the **Default Model Inputs** node on its complementary selection, that is, all domains except those from the **Selection** list. It allows to couple multiple fluid flow interfaces with a single heat transfer interface. See Default Model Inputs in the *COMSOL Multiphysics Reference Manual* for details.

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

The **Nonisothermal Flow** coupling node selection is locked to the union of

- the intersection of the **Fluid** or **Moist Air** features' selection with the **Fluid Properties** feature's selection
- the intersection of the **Porous Medium** feature's selection with the **Fluid and Matrix Properties** feature's selection

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.

HEAT TRANSFER TURBULENCE MODEL

This section is available when the fluid flow interface uses a turbulence model. 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

$$q = -(k + k_T)\nabla T$$

with the turbulent thermal conductivity defined as

$$k_T = \frac{\mu_T C_p}{Pr_T}$$

where μ_T is defined by the flow interface, and Pr_T depends on the **Heat transport turbulence model**. See Turbulent Conductivity for details.

The **Turbulence model type** used by the fluid flow interface can be displayed by selecting the **Show or Hide Physics Property Settings** button at the right of the **Fluid flow** list.

MATERIAL PROPERTIES

Boussinesq Approximation
When the **Compressibility** setting in the fluid flow interface is set to **Incompressible**, select the **Boussinesq approximation** check box in order to use material data evaluated at the reference temperature and reference pressure. If gravity is included in the physics, it is linearized with respect to temperature.

Density
Select an option from the **Specify density** list: **From heat transfer interface** (the default), **From fluid flow interface**, **Custom, linearized density**, or **Custom**:

- For **From heat transfer interface**: define the **Density** ρ in the **Thermodynamics, Fluid** section of the **Fluid** node, in the Heat Transfer coupled interface. Depending on the **Fluid type** option in this node, the density may bet taken from material, set directly, or computed by using the ideal gas law. The same value is automatically set in the **Fluid Properties** section of the **Fluid Properties** node, in the Fluid Flow coupled interface.

- For **From fluid flow interface**: define the **Density** ρ in the **Fluid Properties** section of the **Fluid Properties** node, in the Fluid Flow coupled interface. The same value is automatically set in the **Thermodynamics, Fluid** section of the **Fluid** node, in the Heat Transfer coupled interface.

- For **Custom, linearized density**, enter the **Reference density** ρ_{ref} (SI unit: kg/m^3) and the **Coefficient of thermal expansion** α_p (SI unit:1/K), or select **From material,** or select a variable in the list if available. Regardless how the properties are defined they should be constant. If material properties are not constant you should consider using any of the other options to define the density. Also, if **Custom, linearized density** is used for incompressible flow, the density ρ is evaluated to ρ_{ref}. In this case α_p is not used unless **Boussinesq approximation** is selected. The same value is automatically set in the **Fluid Properties** section of the **Fluid Properties** node, in the Fluid Flow coupled interface, and in the **Thermodynamics, Fluid** section of the **Fluid** node, in the Heat Transfer coupled interface.

- For **Custom**, enter a **Density** ρ (SI unit: kg/m^3), or select a density in the list if available. The same value is automatically set in the **Fluid Properties** section of the **Fluid Properties** node, in the Fluid Flow coupled interface, and in the **Thermodynamics, Fluid** section of the **Fluid** node, in the Heat Transfer coupled interface.

See Fluid Properties and Fluid in the *COMSOL Multiphysics Reference Manual* for details.

When the coupled Heat Transfer interface is also coupled to a Phase Transport interface via the **Nonisothermal Mixture Model** multiphysics coupling, **Specify density** should be set to **From fluid flow interface**. See Nonisothermal Mixture Model in the CFD *Module User's Guide* for details.

The density definition in the **Nonisothermal Flow** node ensures that the same definition of the density is used on the fluid flow and heat transfer interfaces. When the fluid flow compressibility setting is set to **Incompressible** then the thermal conductivity and the heat capacity are evaluated at the **Reference temperature** defined in the fluid flow interface. When **Include gravity** is selected and the **Compressibility** is set to **Incompressible flow** in the fluid interface properties, the gravity forces are defined using the coefficient of thermal expansion. Along with the fact that the material properties are evaluated for a constant temperature and pressure, this gravity force definition corresponds to Boussinesq approximation. Unless the density is defined as **Custom, linearized density** the coefficient of thermal expansion is evaluated from the fluid density.

Reference Temperature

Select an option from the **Specify reference temperature** list — **From heat transfer interface**, **From fluid flow interface** (the default), or **User defined**:

- For **From heat transfer interface**, set the **Reference temperature** T_{ref} (SI unit: K) in the **Physical Model** section of the interface selected in the **Heat transfer** list of the **Coupled Interfaces** section. The **Reference temperature** input in the **Fluid flow** interface is synchronized to the same value or expression, and is not editable.

- For **From fluid flow interface**, set the **Reference temperature** T_{ref} (SI unit: K) in the **Physical Model** section of the interface selected in the **Fluid flow** list of the **Coupled Interfaces** section. The **Reference temperature** input in the **Heat transfer** interface is synchronized to the same value or expression, and is not editable.

- For **User defined**, set a value or expression. The **Reference temperature** inputs in the **Physical Model** sections of the **Heat transfer** and **Fluid flow** interfaces are synchronized to the same value or expression, and are not editable.

When the coupled Heat Transfer interface is also coupled to a Phase Transport interface via the **Nonisothermal Mixture Model** multiphysics coupling, **Specify reference temperature** should be set to **From fluid flow interface**. See Nonisothermal Mixture Model in the *CFD Module User's Guide* for details.

FLOW HEATING

The **Include viscous dissipation** check box is selected by default to account for the heat source corresponding to viscous heating. 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.

> ⚠️ When an interface is selected from the **Heat transfer** list, some of its model inputs are forced with values from the **Nonisothermal 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 fluid flow interface should be used in at most one **Nonisothermal Flow** node. In cases where multiple fluid flow interfaces are used, they can be coupled with a single heat transfer interface, using an equal number of **Nonisothermal Flow** nodes to define proper multiphysics couplings. In the **Heat transfer** interface, the inputs for pressure and velocity are then taken from the first **Nonisothermal Flow** node.

	If a physics interface is deleted and then added to the model again, then in order to reestablish 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 Workflow 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>Nonisothermal 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

Thermal Expansion

Use the **Thermal Expansion** multiphysics coupling (▄) to add an internal thermal strain caused by changes in temperature and account for the corresponding mechanical losses in the heat balance.

See Thermal Expansion 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_Cooling/heating_circuit**

Thermoelectric Effect

Use the **Thermoelectric Effect** multiphysics coupling (▦) to account for a Peltier heat source or sink in domains and on boundaries where electrical and thermal models are defined.

In domains, a $P\mathbf{J}$ contribution is added to the heat flux $\mathbf{q}$ in the equation for heat transfer in solids:

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

The term $\mathbf{J}_e = -\sigma S \nabla T$ is also added 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). See

On boundaries, a $P_s\mathbf{J}$ contribution is added to the heat flux $\mathbf{q}$ in the equation for heat transfer in shells:

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

The term $\mathbf{J}_{e,s} = -\sigma_s S_s \nabla_{\mathbf{t}} T$ is also added to the current density, which is then defined as:

$$\mathbf{J} = -\sigma_s (\nabla_{\mathbf{t}} V + S_s \nabla_{\mathbf{t}} T)$$

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.

DOMAIN SELECTION

From the **Selection** list, choose the domains where thermoelectric effect should be applied. Only domains where both electrical and thermal models are active can be selected.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries where thermoelectric effect should be applied. Only boundaries where both electrical and thermal models are active can be selected.

For the thermal model, you can choose from

- any version of the Heat Transfer interface with a **Thin Layer**, a **Thin Film**, or a **Fracture** feature active on the boundaries.
- any version of the Heat Transfer in Shells interface.

For the electrical model, you can choose from

- any version of the Electric Currents interface with an **Electric Shielding** feature active on the boundaries.
- any version of the Electric Currents, Shells interface.

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.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the coupled heat transfer 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), **Common model input** (the minput.T variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the minput.T variable, click the **Go to Source** button ( ), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate system**, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

See Coordinate Systems in the *COMSOL Multiphysics Reference Manual* for more details.

THERMOELECTRIC PROPERTIES

The Seebeck coefficients S and S_s (SI unit: V/K) in the domain and on the boundaries should be set.

▥	*Thermoelectric Leg*: Application Library path **Heat_Transfer_Module/ Verification_Examples/thermoelectric_leg**

⌕	Theory for the Thermoelectric Effect Interface

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 coupling available:

- Electromagnetic Heating, Layered Shell
- Heat Transfer with Surface-to-Surface Radiation
- Layered Thermal Expansion
- Marangoni Effect
- Thermoelectric Effect, Layered Shell

🔍	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

Electromagnetic Heating, Layered Shell

Use the **Electromagnetic Heating, Layered Shell** multiphysics coupling (🔥) to account for electromagnetic surface losses as a heat source in the heat equation, in layered materials represented by boundaries.

BOUNDARY SELECTION
Select the boundaries on which to define the electromagnetic heat source. The **Restrict to layered boundaries** check box makes the node applicable only if a layered material is defined on the boundary.

COUPLED INTERFACES
This section defines the physics involved in the electromagnetic heating multiphysics coupling.

Select the **Electromagnetic** interface associated to the electric potential dependent variable.

Select the **Heat Transfer** interface associated to the temperature dependent variable.

SHELL PROPERTIES
You can limit the coupling to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer cross section preview** and **Layer 3D preview** buttons.

ℚ	• See Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by explicitly or implicitly using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

ℚ	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

ℚ	See Electromagnetic Heating in the *COMSOL Multiphysics Reference Manual* for a description of the corresponding multiphysics coupling in domains, in the context of Joule heating and laser heating modeling.

LOCATION IN USER INTERFACE

Context Menus

Multiphysics>Electromagnetic Heating, Layered Shell

when any version of the Heat Transfer interface with the **Thin Layer**, **Thin Film**, or **Fracture** node, or any version of the Heat Transfer in Shells interface with the **Solid**, **Fluid**, **Porous Medium** node, is added together with the Electric Currents, Layered Shell interface.

Heat Transfer with Surface-to-Surface Radiation

Use the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling (🔆) to account for heat transfer by radiation on boundaries, when the surrounding medium does not participate in the radiation (transparent medium).

The radiosity equation defined on boundaries where surface-to-surface radiation is enabled corresponds to the radiosity method equation.

The following radiative heat source is added to the heat transfer equation on boundaries:

$$q = \varepsilon(G - e_b(T))$$

on the side of the boundary where the radiation is defined, where ε is the surface emissivity, G is the irradiation, and $e_b(T)$ 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.

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 htrad1.

COUPLED INTERFACES

This section defines the physics involved in the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling.

Select the **Heat transfer** interface associated to the temperature dependent variable. Select the **Surface-to-surface radiation** interface associated to the radiosity variable.

> In 1D, 1D-axi, and 2D, when the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling is active while solving, it disables all the out of plane features (Out-of-Plane Heat Flux, Out-of-Plane Radiation, Thickness, Cross Section) from the heat transfer interface that is selected in the multiphysics coupling. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION

Set the default opacities of the domains surrounding the boundaries where radiation is modeled.

If **Default domain opacities** is **From heat transfer interface**, the opacity depends on the domain feature:

- The **Solid**, **Porous Medium**, **Biological Tissue**, **Building Material**, and **Shape Memory Alloy** nodes define opaque domains.

- The **Fluid** and **Isothermal Domain** nodes define transparent domains.

Note that when a Heat Transfer in Shells interface is selected under **Heat transfer** in the **Coupled Interfaces** section, no domain node is available. Therefore, the default opacity set in the Surface-to-Surface Radiation interface is used: all surrounding domains are considered as transparent.

If **Default domain opacities** is **Transparent**, all domains are considered transparent by default.

📄	With either option the default opacity may be overriden on each domain with the Opacity (Surface-to-Surface Radiation Interface) boundary node.

📄	The opacity setting is used when the **Emitted radiation direction** is defined by **Opacity controlled** in surface-to-surface boundary features (Diffuse Mirror (Surface-to-Surface Radiation Interface), Diffuse Surface (Surface-to-Surface Radiation Interface), Prescribed Radiosity (Surface-to-Surface Radiation Interface), and Opaque Surface (Surface-to-Surface Radiation Interface)): surface-to-surface radiation propagates in non-opaque domains. Alternatively the **Emitted radiation direction** can be defined using the normal orientation or on both sides of boundaries. In this case this setting is ignored. Note that on boundaries where the Semitransparent Surface (Surface-to-Surface Radiation Interface) is applied, the **Emitted radiation direction** is **Both sides**, and this setting is always ignored.

Context Menus

Multiphysics>Heat Transfer with Surface-to-Surface Radiation

when a Heat Transfer interface, a Heat Transfer in Shells interface, a Slip Flow interface, or a High Mach Number Flow interface is added together with the Surface-to-Surface Radiation interface.

Layered Thermal Expansion

Use the **Layered Thermal Expansion** multiphysics coupling (▤) to add an internal thermal strain caused by changes in temperature and account for the corresponding mechanical losses in the heat balance in layered materials represented by boundaries.

See Layered Thermal Expansion in the *Structural Mechanics Module User's Guide* for more details about this multiphysics coupling.

Marangoni Effect

The **Marangoni Effect** multiphysics coupling (🔴) accounts for Marangoni convection. Marangoni convection occurs when the surface tension of a fluid-fluid 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 that depends on the tangential and normal variations of surface tension with temperature gradient. It has the following contribution described by forces induced on the fluid/fluid interface:

$$\left[-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} \right]\mathbf{n} = \sigma(\nabla_t \cdot \mathbf{n})\mathbf{n} - \nabla_t\sigma$$

where σ is the surface tension coefficient (N/m). The first term on the right-hand-side accounts for the normal forces related to curvature effects, while the second term stands for tangential forces. 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 mar1.

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.

> 📄 If a physics interface is deleted and then added to the model again, then in order to reestablish 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 Workflow in the *COMSOL Multiphysics Reference Manual*.

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).

The attachment angle between the fluid/fluid interface and the adjacent walls can be specified in this section.

Select an option from the **Specify contact angle** list — **Directly** (the default) or **Through Young's equation**.

- For **Directly** enter a **Contact angle** θ_w (SI unit: rad). The default is $\pi/2$ radians.

- For **Through Young's equation** enter values or expressions for **Phase 1-Solid surface energy density** γ_{s1} (SI unit: J/m^2) and **Phase 2-Solid surface energy density** γ_{s2} (SI unit: J/m^2).

▥	*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 Fluids** (or another version of the Heat Transfer Interface):

Single-Phase Flow (any version)
Porous Media and Subsurface Flow, Brinkman Equations

Thermoelectric Effect, Layered Shell

Use the **Thermoelectric Effect, Layered Shell** multiphysics coupling (▦) to account for a Peltier heat source or sink in layered materials represented by boundaries where electrical and thermal models are defined.

In the equation for heat transfer in shells, a $q_s = P_s \mathbf{J}$ contribution is added to the heat flux $\mathbf{q}$:

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

The term $\mathbf{J}_{e,s} = -\sigma_s S_s \nabla_{\mathbf{t}} T$ is also added to the current density, which is then defined as:

$$\mathbf{J} = -\sigma_s(\nabla_{\mathbf{t}}V + S_s\nabla_{\mathbf{t}}T)$$

BOUNDARY SELECTION

Select the boundaries on which to define the electromagnetic heat source. The **Restrict to layered boundaries** check box makes the node applicable only if a layered material is defined on the boundary.

COUPLED INTERFACES

This section defines the physics involved in the electromagnetic heating multiphysics coupling.

Select the **Electromagnetic** interface associated to the electric potential dependent variable.

Select the **Heat Transfer** interface associated to the temperature dependent variable.

SHELL PROPERTIES

You can limit the coupling to individually selected layers by clearing the **Use all layers** check box. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of check boxes for the selection of the individual layers.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer cross section preview** and **Layer 3D preview** buttons.

Q	• See Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by explicitly or implicitly using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.

Q	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the *COMSOL Multiphysics Reference Manual* for details on the definition of layered materials.

> See Electromagnetic Heating in the *COMSOL Multiphysics Reference Manual* for a description of the corresponding multiphysics coupling in domains, in the context of Joule heating and laser heating modeling.

THERMOELECTRIC PROPERTIES

The **Seebeck coefficient** S_s (SI unit: V/K) on the boundaries should be set. By default, it is taken **From layered material**. For **User defined**, it should be specified.

LOCATION IN USER INTERFACE

Context Menus

Multiphysics>Thermoelectric Effect, Layered Shell

when any version of the Heat Transfer interface with the **Thin Layer**, **Thin Film**, or **Fracture** node, or any version of the Heat Transfer in Shells interface with the **Solid**, **Fluid**, **Porous Medium** node, is added together with the Electric Currents, Layered Shell interface.

Index