

Heat Transfer Module

User's Guide

Heat Transfer Module User's Guide

© 1998–2020 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 9,098,106; 9,146,652; 9,323,503; 9,372,673; 9,454,625; 10,019,544; 10,650,177; and 10,776,541. Patents pending.

This Documentation and the Programs described herein are furnished under the COMSOL Software License Agreement (www.comsol.com/comsol-license-agreement) and may be used or copied only under the terms of the license agreement.

COMSOL, the COMSOL logo, COMSOL Multiphysics, COMSOL Desktop, COMSOL Compiler, COMSOL Server, and LiveLink are either registered trademarks or trademarks of COMSOL AB. All other trademarks are the property of their respective owners, and COMSOL AB and its subsidiaries and products are not affiliated with, endorsed by, sponsored by, or supported by those trademark owners. For a list of such trademark owners, see www.comsol.com/trademarks.

Version: COMSOL 5.6

Contact Information

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

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

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

Part number: CM020801

C o n t e n t s

Chapter 1: Introduction

About the Heat Transfer Module	22
Why Heat Transfer Is Important to Modeling	22
How the Heat Transfer Module Improves Your Modeling	23
The Heat Transfer Module Physics Interface Guide	23
Common Physics Interface and Feature Settings and Nodes	31
The Heat Transfer Module Study Capabilities	31
Additional Materials Database	36
Where Do I Access the Documentation and Application Libraries?	36
Overview of the User's Guide	40

Chapter 2: Notations

Symbols	46
----------------	-----------

Chapter 3: Modeling with the Heat Transfer Module

Heat Transfer Variables	62
Predefined Variables	62
Global Variables for Heat and Energy Balance	66
Domain Fluxes	69
Out-of-Plane Domain Fluxes	71
Boundary Fluxes (Heat Transfer Interface)	72
Internal Boundary Heat Fluxes	73
Internal Boundary Energy Fluxes	75
Domain Heat Sources	75
Boundary Heat Sources	76
Line and Point Heat Sources	76
Moist Air Variables	76

Moisture Transport Variables	79
Predefined Variables	79
Moist Air Properties	80
Domain Moisture Fluxes	82
Boundary Moisture Fluxes	83
Domain Moisture Source.	84
Using the Boundary Conditions for the Heat Transfer	
Interfaces	85
Temperature and Heat Flux Boundary Conditions	85
Overriding Mechanism for Heat Transfer Boundary Conditions	86
Handling Frames in Heat Transfer	89
Heat Transfer Analysis with Moving Frames.	89
Material Density in Features Defined in the Material Frame.	94
Heat Transfer Consistent and Inconsistent Stabilization	
Methods	95
Consistent Stabilization	95
Inconsistent Stabilization	96
Heat Transfer and Fluid Flow Coupling	97
Coupling Based on Model Inputs	97
Adding Nonisothermal Flow Coupling in an Existing Model	98
Nonisothermal Flow and Conjugate Heat Transfer Multiphysics	
Interfaces	98
Boundary Wall Temperature	100
Solver Settings	105
Linearity Property of the Temperature Equation	105
Linear Solver	106
Nonlinear Solver	107
Time-Dependent Study Step	111
Guidelines for Solving Surface-to-Surface Radiation Problems	112
Guidelines for Solving Multiphysics Problems	113

Plotting and Evaluating Results in Layered Materials	116
Plotting Along and Through the Layered Material	116
Built-in Operators for Evaluation in the Layered Material	117
Ambient Properties	121
Settings for the Ambient Properties.	121
Processing of ASHRAE Data	125
Ambient Data Interpolation	126
Ambient Variables and Conditions	127
Use of Ambient Data From the Features	135
Modeling Heat Transfer by Radiation	137
Interfaces and Features for Radiation in Transparent Media.	139
Interfaces and Features for Radiation in Participating Media.	141
Specifying the Temperature at a Flow Inlet	144
Using the Temperature Condition	144
Using the Inflow Condition	144
Temperature Contribution to the Inflow Boundary Condition.	146
Pressure Contribution to the Inflow Boundary Condition	146
Choosing Between the Temperature and the Inflow Conditions	147
Heat Part Library	148
Heat Part Library Contents	148
Heat Sink — Parameterized Fin Types Part	151
Heat Sink — Dissimilar Border Pins Part	154
Heat Sink — Pin Fins Part	154
Heat Sink — Straight Fins Part.	154
References	155

Chapter 4: Theory for the Heat Transfer Module

Foundations of the General Heat Transfer Equation	159
Thermodynamic Description of Heat Transfer.	159
The Physical Mechanisms Underlying Heat Transfer.	164

The Heat Balance Equation	165
Consistency with Mass and Momentum Conservation Laws.	168
Theory for Heat Transfer in Solids	170
Theory for Heat Transfer in Fluids	171
Theory for the Inflow Boundary Condition	172
Theory for Heat Transfer in Moist Air	174
Humidity	174
Saturation State	176
Moist Air Properties	176
Theory for Bioheat Transfer	181
The Bioheat Equation	181
Damaged Tissue	181
Theory for Heat Transfer in Porous Media	185
When Should Local Thermal Equilibrium and Nonequilibrium Be Considered?	185
Local Thermal Equilibrium	186
Local Thermal Nonequilibrium	188
Theory for Heat Transfer in Moist Porous Media	190
Theory for Heat Transfer with Phase Change	193
Heat Transfer with Phase Change	193
Stefan Condition	193
Apparent Heat Capacity Method	194
Theory for Heat Transfer in Building Materials	198
Theory for Harmonic Heat Transfer	200
Theory for Lumped Isothermal Domain	202
Theory for Heat Transfer in Thin Structures	205
Modeling Layered Materials	205
Theoretical Background of the Different Formulations	210

Thin Layer	215
Thin Film	218
Fracture	220
Thin Rod	221
Theory for the Lumped Thermal System Interface	222
Temperature Differences and Heat Rates in the Lumped Thermal System	222
Theory for the Conductive Thermal Resistor Component	224
Theory for the Radiative Thermal Resistor Component	228
Theory for the Thermal Capacitor and Thermal Mass Components	230
Theory for the Heat Pipe Component.	231
Theory for the Thermoelectric Module Component	233
Theory for Surface-to-Surface Radiation	237
Deriving the Radiative Heat Flux for Opaque Surfaces.	237
Deriving the Radiative Heat Flux for Semitransparent Surfaces	238
Wavelength Dependence of Surface Emissivity and Absorptivity	240
Directional Dependence of Surface Properties	246
The Radiosity Method for Diffuse-Gray Surfaces	247
The Radiosity Method for Diffuse-Spectral Surfaces.	250
View Factor Evaluation	253
Theory for Radiation in Participating Media	258
Radiation and Participating Media Interactions	258
Radiative Transfer Equation	259
Boundary Condition for the Radiative Transfer Equation.	261
Heat Transfer Equation in Participating Media	263
Discrete Ordinates Method (DOM)	264
Discrete Ordinates Method Implementation in 2D	266
PI Approximation Theory	268
Radiation in Absorbing-Scattering Media Theory	271
Polychromatic Radiation	273
Radiative Beam in Absorbing Media Theory.	273
Rosseland Approximation Theory	274
Theory for Moisture Transport	276
Theory for Moisture Transport in Air	276

Theory for Moisture Transport in Porous Media	277
Theory for Moisture Transport in Building Materials	280
Theory for the Heat Transfer Multiphysics Couplings	283
Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces	283
Theory for the Moisture Flow Interface	291
Theory for the Thermoelectric Effect Interface	295
Theory for the Local Thermal Nonequilibrium Interface	298
Theory for the Building Materials Version of the Heat and Moisture Transport Interface	299
Theory for the Moist Air Version of the Heat and Moisture Transport Interface	300
Theory for the Moist Porous Media Version of the Heat and Moisture Transport Interface	301
Theory for the Heat and Moisture Flow Interfaces	302
Theory for the Electromagnetic Heating Interfaces	302
Theory for the Thermal Stress Interface	302
Theory for Thermal Contact	303
Theory for the Thermal Contact Feature	303
Out-of-Plane Heat Transfer	308
Equation Formulation	308
The Heat Transfer Coefficients	311
Defining the Heat Transfer Coefficients	312
Nature of the Flow — The Grashof Number	313
Heat Transfer Coefficients — External Natural Convection	315
Heat Transfer Coefficients — Internal Natural Convection	322
Heat Transfer Coefficients — External Forced Convection	323
Heat Transfer Coefficients — Internal Forced Convection	326
Using the Heat and Mass Transfer Analogy for the Evaluation of Moisture Transfer Coefficients	326
Equivalent Thermal Conductivity Correlations	328
Horizontal Cavity With Bottom Heating	329
Vertical Cavity With Sidewall Heating	329

Temperature Dependence of Surface Tension	331
Heat Flux and Heat Balance	332
Total Heat Flux and Energy Flux	332
Heat and Energy Balance	333
Frames for the Heat Transfer Equations	336
Material and Spatial Frames	336
Conversion Between Material and Spatial Frames	337
References	341

Chapter 5: The Heat Transfer Module Interfaces

About the Heat Transfer Interfaces	348
Space Dimensions	348
Study Types	349
Versions of the Heat Transfer Physics Interface	349
Benefits of the Different Heat Transfer Interfaces	350
Versions of the Heat Transfer in Shells Physics Interface	350
Benefits of the Different Heat Transfer in Shells Interfaces	351
Layer and Interface Selection Tools	351
Using the Extra Dimension Coordinates	355
Versions of the Moisture Transport Physics Interface	355
Benefits of the Different Moisture Transport Interfaces	356
Additional Physics Options	356
Settings for the Heat Transfer Interface	357
Feature Nodes for the Heat Transfer Interface	360
Settings for the Heat Transfer in Shells Interface	363
Settings for the Moisture Transport Interface	368
Feature Nodes for the Moisture Transport Interface	370

The Heat Transfer in Solids Interface	371
The Heat Transfer in Fluids Interface	372
The Heat Transfer in Solids and Fluids Interface	373
The Heat Transfer in Porous Media Interface	374
The Heat Transfer in Moist Porous Media Interface	375
The Heat Transfer in Building Materials Interface	376
The Heat Transfer in Moist Air Interface	377
The Bioheat Transfer Interface	378
The Heat Transfer in Shells Interface	379
Feature Nodes for the Heat Transfer in Shells Interface	381
The Heat Transfer in Films Interface	383
Feature Nodes for the Heat Transfer in Films Interface	385
The Heat Transfer in Fractures Interface	387
Feature Nodes for the Heat Transfer in Fractures Interface	389
The Lumped Thermal System Interface	392
Feature Nodes for the Lumped Thermal System Interface	393
The Surface-to-Surface Radiation Interface	395
Settings for the Surface-to-Surface Radiation Interface.	395
Feature Nodes for the Surface-to-Surface Radiation Interface	399
The Radiation in Participating Media Interface	401
Settings for the Radiation in Participating Media Interface	401
Feature Nodes for the Radiation in Participating Media Interface.	404
The Radiation in Absorbing-Scattering Media Interface	406
Settings for the Radiation in Absorbing-Scattering Media Interface	406

Feature Nodes for the Radiation in Absorbing-Scattering Media Interface	409
The Radiative Beam in Absorbing Media Interface	411
Settings for the Radiative Beam in Absorbing Media Interface	411
Feature Nodes for the Radiative Beam in Absorbing Media Interface	412
The Moisture Transport in Air Interface	414
The Moisture Transport in Porous Media Interface	415
The Moisture Transport in Building Materials Interface	416

Chapter 6: The Heat Transfer Features

Domain Features	418
Absorbing Medium (Radiative Beam in Absorbing Medium Interface)	419
Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface)	420
Bioheat	425
Biological Tissue	426
Building Material	429
Convectively Enhanced Conductivity	434
Cross Section	435
Fluid	436
Fluid (Porous Medium)	441
Geothermal Heating	445
Heat Source	446
Immobile Fluids (Porous Medium)	449
Initial Values	452
Initial Values (Radiative Beam in Absorbing Medium Interface)	453
Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	453
Irreversible Transformation	456
Isothermal Domain	459
Liquid Water (Moist Porous Medium)	461

Moist Air (Heat Transfer Interface)	462
Moist Air (Moist Porous Medium)	464
Moist Porous Medium	465
Opacity (Surface-to-Surface Radiation Interface)	468
Optically Thick Participating Medium	469
Out-of-Plane Heat Flux	471
Out-of-Plane Radiation	473
Participating Medium (Radiation in Participating Media Interface).	475
Phase Change Material.	480
Porous Matrix (Porous Medium, Moist Porous Medium)	484
Porous Medium	488
Pressure Work	491
Radiative Source	492
Shape Memory Alloy	494
Solid	498
Thermal Damage	501
Thermal Dispersion.	505
Thermoelastic Damping	506
Thickness.	507
Translational Motion	508
Viscous Dissipation	509
Boundary Features	511
Boundary Heat Source.	513
Continuity	516
Continuity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	517
Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	517
Deposited Beam Power	518
Diffuse Mirror (Surface-to-Surface Radiation Interface)	519
Diffuse Surface (Surface-to-Surface Radiation Interface)	521
External Temperature (Thin Layer, Thin Film, Fracture)	527
Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)	528
Harmonic Perturbation	533
Heat Flux.	534
Heat Source (Heat Transfer in Shells Interface)	538

Heat Source (Thin Layer, Thin Film, Fracture)	540
Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	543
Incident Intensity (Radiative Beam in Absorbing Medium Interface)	545
Inflow	547
Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	548
Initial Values (Surface-to-Surface Radiation Interface)	550
Isothermal Domain Interface	552
Layer Opacity (Surface-to-Surface Radiation Interface)	556
Line Heat Source on Axis	557
Opaque Surface (Surface-to-Surface Radiation Interface)	557
Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	561
Opaque Surface (Radiative Beam in Absorbing Medium Interface)	564
Lumped System Connector	565
Open Boundary	566
Outflow	567
Periodic Condition (Heat Transfer Interface)	568
Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	569
Phase Change Interface	570
Prescribed Radiosity (Surface-to-Surface Radiation Interface)	573
Radiation Group (Surface-to-Surface Radiation Interface)	577
Semitransparent Surface (Radiation in Participating Media Interface)	580
Semitransparent Surface (Surface-to-Surface Radiation Interface).	585
Surface-to-Ambient Radiation (Heat Transfer Interface)	589
Symmetry (Heat Transfer Interface).	591
Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	591
Temperature	592
Thermal Contact	594
Thermal Insulation	599
Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)	599
Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)	604
Transparent Surface (Radiative Beam in Absorbing Medium Interface).	609

Boundary Interface Features	611
Deposited Beam Power, Interface (Heat Transfer in Shells Interface) . . .	611
Heat Flux, Interface (Heat Transfer in Shells Interface)	614
Heat Source, Interface (Heat Transfer in Shells Interface)	617
Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)	620
Temperature, Interface (Heat Transfer in Shells Interface)	623
Thermal Contact, Interface (Heat Transfer in Shells Interface).	625
 Edge Features	 629
Heat Flux (Heat Transfer in Shells Interface)	629
Heat Flux (Thin Layer, Thin Film, Fracture)	632
Heat Source (Heat Transfer in Shells Interface)	635
Line Heat Source.	638
Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)	640
Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	641
Symmetry (Heat Transfer in Shells Interface)	644
Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	645
Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	648
Thin Rod	650
 Point Features	 652
Point Heat Flux (Thin Rod)	652
Point Heat Source	653
Point Heat Source on Axis	654
Surface-to-Ambient Radiation (Thin Rod)	655
Temperature (Thin Rod)	656
 Global Features	 658
Conductive Thermal Resistor	658
Convective Thermal Resistor	662
External Radiation Source	664
Symmetry for Surface-to-Surface Radiation	670
External Terminal	673

Heat Pipe	674
Heat Rate	678
Heat Rate Source	680
Radiative Heat Rate	681
Radiative Thermal Resistor	683
Subsystem Definition	684
Subsystem Instance	685
Temperature	686
Thermal Capacitor	686
Thermal Mass	688
Thermoelectric Module	689

Chapter 7: The Moisture Transport Features

Domain Features	696
Building Material	696
Hygroscopic Porous Medium	699
Initial Values	702
Moist Air (Moisture Transport Interface).	702
Moisture Source	705
Turbulent Mixing	706
Boundary Features	707
Continuity	707
Inflow	708
Insulation.	709
Moist Surface	710
Moisture Content	712
Moisture Flux	713
Open Boundary	715
Outflow	717
Symmetry	718
Thin Moisture Barrier	718
Wet Surface	719

Global Features	722
Gravity	722

Chapter 8: Multiphysics Interfaces

The Nonisothermal Flow and Conjugate Heat Transfer	
Interfaces	726
Advantages of Using the Multiphysics Interfaces	726
The Nonisothermal Flow, Laminar Flow and Turbulent Flow	
Interfaces	727
The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow	
Interfaces	728
Settings for Physics Interfaces and Coupling Features	729
Coupling Feature	730
Physics Interface Features	730
Preset Studies	731
The Heat Transfer with Surface-to-Surface Radiation Interface	733
The Heat Transfer with Surface-to-Surface Radiation Multiphysics	
Interface	733
Physics Interface Features	735
Coupling Feature	735
The Heat Transfer with Radiation in Participating Media	
Interface	736
The Heat Transfer with Radiation in Participating Media Multiphysics	
Interface	736
Physics Interface Features	737
Coupling Feature	738
The Heat Transfer with Radiation in Absorbing-Scattering	
Media Interface	739
The Heat Transfer with Radiation in Absorbing-Scattering Media	
Multiphysics Interface	739
Physics Interface Features	740

Coupling Feature	741
The Heat Transfer with Radiative Beam in Absorbing Media Interface	742
The Heat Transfer with Radiative Beam in Absorbing Media	
Multiphysics Interface	742
Physics Interface Features	743
Coupling Feature	743
The Thermoelectric Effect Interface	744
About The Thermoelectric Effect Interface	744
Settings for Physics Interfaces and Coupling Features	745
Coupling Features	746
Physics Interface Features	747
The Local Thermal Nonequilibrium Interface	748
About the Local Thermal Nonequilibrium Interface	748
Coupling Feature	749
Physics Interface Features	749
The Heat and Moisture Transport Interfaces	750
The Heat and Moisture Transport Multiphysics Interfaces	750
The Moist Air Version of the Heat and Moisture Transport	
Multiphysics Interface	751
The Moist Porous Media Version of the Heat and Moisture	
Transport Multiphysics Interface	752
The Building Materials Version of the Heat and Moisture Transport	
Multiphysics Interface	754
Physics Interface Features	756
Coupling Feature	756
The Moisture Flow Interfaces	757
The Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics	
Interfaces	757
Physics Interface Features for the Laminar and Turbulent Flow	
Versions	760
The Moisture Flow, Brinkman Equations Multiphysics Interface	761
Physics Interface Features for the Brinkman Equations Version	762

Preset Studies	762
Coupling Feature.	762
The Heat and Moisture Flow Interfaces	763
The Heat and Moisture Flow, Laminar Flow and Turbulent Flow	
Multiphysics Interfaces	763
Physics Interface Features for the Laminar and Turbulent Flow	
Versions.	767
The Heat and Moisture Flow, Brinkman Equations Interface	768
Physics Interface Features for the Brinkman Equations Version	771
Coupling Features	771
The Joule Heating Interface	772
The Joule Heating Interface	772
Coupling Feature.	772
The Laser Heating Interface	773
The Laser Heating Interface.	773
Coupling Feature.	773
The Induction Heating Interface	774
The Induction Heating Interface	774
Coupling Feature.	774
The Microwave Heating Interface	775
The Microwave Heating Interface	775
Coupling Feature.	775

Chapter 9: Multiphysics Couplings

Domain Multiphysics Couplings	779
Electromagnetic Heating	779
Heat and Moisture	780
Heat Transfer with Radiation in Participating Media.	783
Heat Transfer with Radiation in Absorbing-Scattering Media	784
Heat Transfer with Radiative Beam in Absorbing Media	786

Local Thermal Nonequilibrium	787
Moisture Flow	789
Nonisothermal Flow	792
Thermal Expansion	798
Thermoelectric Effect	798
Boundary Multiphysics Couplings	802
Electromagnetic Heating, Layered Shell	802
Heat Transfer with Surface-to-Surface Radiation	803
Layered Thermal Expansion	805
Marangoni Effect	806
Thermoelectric Effect, Layered Shell	808
Index	811

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 γ Plus ⁽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 γ Plus		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 γ Plus ⁽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, $k-\epsilon$ ⁽²⁾		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, Low Re $k-\epsilon$ ⁽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 γ Plus ⁽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				
Joule Heating ^(1,2)		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Lumped Thermal System		lts	all dimensions	stationary; time dependent
 Thin Structures				
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 γ Plus ⁽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, $k-\epsilon$ ⁽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 $k-\epsilon$ ⁽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
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Local Thermal Nonequilibrium ⁽²⁾		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency
Bioheat Transfer		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Thermoelectric Effect ⁽²⁾		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
<p>(1) This physics interface is included with the core COMSOL package but has added functionality for this module.</p> <p>(2) This physics interface is a predefined multiphysics coupling that automatically adds all the physics interfaces and coupling features required.</p>				

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

PHYSICS INTERFACE	NAME	DEPENDENT VARIABLES	PRESET STUDIES ¹					
			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 <i>yPlus</i> ⁽²⁾	—	$u, p, G, yPlus, \phi$					√	√
Turbulent flow, L-VEL ⁽²⁾	—	$u, p, G, uPlus, \phi$					√	√
Turbulent Flow, $k-\epsilon$ ⁽²⁾	—	u, p, k, ep, ϕ	√	√				
Turbulent Flow, Low Re $k-\epsilon$ ⁽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 <i>yPlus</i>	spf	$u, p, G, yPlus$					√	√
Turbulent flow, L-VEL	spf	$u, p, G, uPlus$					√	√
Turbulent Flow, $k-\epsilon$	spf	u, p, k, ep	√	√				

TABLE I-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 γ Plus ⁽²⁾	—	$u, p, G, \gamma Plus, T$					√	√
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 γ Plus ⁽²⁾	—	$u, p, G, \gamma Plus, T$					√	√
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 I-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
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 STRUCTURES								
Heat Transfer in Shells	htlsh	T	√	√	√	√		
Heat Transfer in Films	htlsh	T	√	√	√	√		
Heat Transfer in Fractures	htlsh	T	√	√	√	√		
HEAT TRANSFER>HEAT AND MOISTURE TRANSPORT								

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

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, $k-\epsilon$ ⁽²⁾	—	u, p, k, ep, T, ϕ	√	√				
Turbulent Flow, Low Re $k-\epsilon$ ⁽²⁾	—	u, p, k, ep, G, T, ϕ					√	√
Porous Media Flow ⁽²⁾	—	u, p, T, ϕ	√	√				
Heat Transfer in Porous Media	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.


Opening Topic-Based Help

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 | <p>To open the Help window:</p> <ul style="list-style-type: none">• 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 | <p>To open the Help window:</p> <ul style="list-style-type: none">• 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. |
| Linux | |
-

Opening the Documentation Window

-
- | | |
|-----|---|
| Win | <p>To open the Documentation window:</p> <ul style="list-style-type: none">• Press Ctrl+F1.• From the File menu select Help>Documentation (). |
|-----|---|
-



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 ():




- 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**.



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](#), [The Heat Transfer Coefficients](#), [Equivalent Thermal Conductivity Correlations](#), [Temperature Dependence of Surface Tension](#), [Heat Flux and Heat Balance](#), and [Frames for the Heat Transfer Equations](#).

THE HEAT TRANSFER MODULE INTERFACES

The 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](#) 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 Local Thermal Nonequilibrium Interface](#) describes the predefined multiphysics interface used to model heat transfer in porous media when there is no thermal equilibrium between porous and fluid phases.

The 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.

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, \dots, a_{12}	dimensionless	Legendre coefficients
A_c	m^2	Cross sectional area of domain
A_l	m^2	Cross sectional area of thin rod
a_{sf}	1/m	Specific surface area
B_i	m	Spectral band i
b	dimensionless	Thermal conductivity supplement
c	mol/m^3	Concentration
c	kg/m^3	Concentration
c_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
$C_{p, imf}$	J/(kg·K)	Specific heat capacity at constant pressure of immobile fluid in porous media
$C_{p, imfi}$	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

NOTATION	SI UNIT	DESCRIPTION
$C_{p,v}$	J/(kg·K)	Specific heat capacity at constant pressure, water vapor
$C_{p,s}$	J/(kg·K)	Specific heat capacity at constant pressure of thin layer
$C_{p,si}$	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
$C_{p,w}$	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_{P1}	m ² /s	PI method diffusion coefficient
d_s	m	Thickness of shell or thin layer
d_{si}	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
DPT_{amb}	K	Ambient dew point temperature
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
e_{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}	$kg/(m^2 \cdot s)$	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/(m ² ·K)	Gap conductance (thermal contact)
h	W/(m ² ·K)	Heat transfer coefficient
H_0	J/kg	Total enthalpy
H_B	Pa	Brinell hardness
h_c	W/(m ² ·K)	Constriction conductance
H_c	Pa	Microhardness
h_d	W/(m ² ·K)	Out-of-plane heat transfer coefficient, downside
H_d	J/kg	Enthalpy, downside
H_{ext}	J/kg	External enthalpy
h_g	W/(m ² ·K)	Parallel-plate gap gas conductance
h_m	m/s	Moisture transfer coefficient
h_r	W/(m ² ·K)	Radiative conductance
H_{ref}	J/kg	Reference enthalpy
h_{sf}	W/(m ² ·K)	Interstitial heat transfer coefficient
h_{th}	W/K	Thermal conductance (isothermal domain interface)
h_u	W/(m ² ·K)	Out-of-plane heat transfer coefficient, upside
H_u	J/kg	Enthalpy, upside
h_z	W/(m ² ·K)	Out-of-plane heat transfer coefficient, ID
$I(\Omega)$	W/(m ² ·sr)	Radiative intensity traveling in direction Ω
$I_b(T)$	W/(m ² ·sr)	Blackbody radiative intensity
I_{diff}	W/m ²	Diffuse irradiance
I_{ext}	W/(m ² ·sr)	Incident radiative intensity
I_i	W/(m ² ·sr)	Radiative intensity traveling in i th discrete direction
\mathbf{i}_s	dimensionless	Incident radiation direction (external radiation source)
I_s	W/m ²	Solar irradiance
$I_{s,amb}$	W/m ²	Ambient solar irradiance
$I_{sh,amb}$	W/m ²	Clear sky noon diffuse horizontal irradiance
$I_{sn,amb}$	W/m ²	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_{imfi}	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_{si}	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_{si}	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
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}_f	W/m ²	Conductive heat flux in fluid phase
\mathbf{q}_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
$q_{0,d}$	W/m ²	Out-of-plane heat flux, downside
$q_{0,s}$	W/m ²	Source heat flux (external radiation source)
$q_{0,u}$	W/m ²	Out-of-plane heat flux, upside
Q_b	W/m ²	Boundary heat source
$Q_{b,tot}$	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
q_{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
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
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_p	m	Average pellet 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_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, ID
T_f	K	Temperature, fluid phase
$t_{it,c}$		Transformation time, overcooling analysis

NOTATION	SI UNIT	DESCRIPTION
$T_{it, c}$		Transformation temperature, overcooling analysis
$t_{it, h}$		Transformation time, overheating analysis
$T_{it, h}$		Transformation temperature, overheating analysis
$T_{n, c}$	K	Temperature of necrosis, cryogenic
$T_{n, h}$	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}_f	m/s	Average linear velocity
\mathbf{u}_p	m/s	Porous velocity field
\mathbf{u}_{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}_s	m	Source location (external radiation source)
X_v	dimensionless	Molar fraction of water vapor
x_{vap}	dimensionless	Moisture content
$x_{vap, amb}$	dimensionless	Ambient moisture content
Y	m	Parallel-plate gap mean separation thickness

GREEK SYMBOLS

NOTATION	SI UNIT	DESCRIPTION
$(\rho C_p)_{\text{eff}}$	$\text{J}/(\text{m}^3 \cdot \text{K})$	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	$1/\text{K}$	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	$1/\text{K}$	Coefficient of thermal expansion in a fluid
β	$1/\text{m}$	Extinction coefficient
β_p	s/m	Moisture transfer coefficient
β	dimensionless	Parallel-plate gap gas property parameter
β_{R}	$1/\text{m}$	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_{10}	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	l/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
$\Phi_{d \rightarrow u}$	kg/s	Mass flow rate, positive direction
$\Phi_{u \rightarrow d}$	kg/s	Mass flow rate, negative direction
$\varphi(\Omega', \Omega)$	dimensionless	Scattering phase function
φ_s	rad	Azimuth angle of the Sun
$\varphi_{d \rightarrow u}$	kg/(m ² ·s)	Mass flux, positive direction
$\varphi_{u \rightarrow d}$	kg/(m ² ·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	l/s	Blood perfusion rate
ω_i	W/(m ² ·K)	Discrete incident radiation vector, <i>i</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](#)
- [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 `ht1sh`. 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		
<code>dEiInt</code>	Total Accumulated Heat Rate	Global
<code>ntfluxInt</code>	Total Net Heat Rate	Global
<code>WstrInt</code>	Total Stress Power	Global
Domain variables		
<code>tflux</code>	Total Heat Flux (Heat Transfer Interface)	Domains
<code>dflux</code>	Conductive Heat Flux (Heat Transfer Interface)	Domains

TABLE 3-1: PREDEFINED HEAT BALANCE VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
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
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 The Heat Transfer Coefficients)	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, $\partial\Omega_{\text{ext}}$ stands for the exterior boundaries, and $\partial\Omega_{\text{int}}$ 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:

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

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:

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

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:

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

TOTAL STRESS POWER

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

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

TOTAL ACCUMULATED ENERGY RATE

The total accumulated energy rate, dE_{i0Int} , is the variation of total internal energy per unit time in the domain:

$$dE_{i0Int} = \frac{d}{dt} \int_{\Omega} \rho E_{i0} d\omega$$

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, $ntfluxInt$, is the integral of [Total Energy Flux \(Heat Transfer Interface\)](#) over all external boundaries. In the case of a fluid domain, it reads:

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

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:

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

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

HEAT BALANCE

According to [Equation 4-198](#), the following equality between COMSOL Multiphysics variables holds:

$$dE_{i0Int} + ntfluxInt = QInt - WstrInt$$

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:

$$\text{ntfluxInt} = \text{QInt} - \text{WstrInt}$$

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:

$$\text{ntfluxInt} = \text{QInt}$$

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 **PI approximation** and **Discrete ordinates method**, however, include the radiative source $\nabla \cdot \mathbf{q}_r$ to Q on the domain, in the variable **QInt**.

ENERGY BALANCE

According to [Equation 4-199](#), the following equality between COMSOL Multiphysics predefined variables holds:

$$\text{dEi0Int} + \text{ntefluxInt} = \text{QInt} + \text{WInt}$$

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

$$\text{ntefluxInt} = \text{QInt} + \text{WInt}$$

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.s1s1.ntfluxInt`, defined locally on the thin structure. Here, `s1s1` is the tag of the **Thin Layer** node.

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:

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

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

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

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

When considering thermoelectric effect, the thermoelectric heat flux is also included in the diffusive flux, see [Equation 4-148](#).

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 :

$$\text{cflux} = \rho\mathbf{u}E_i$$

The internal energy, E_i , is defined as:

- $E_i = H$ for solid domains
- $E_i = H - p/\rho$ 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/m³), 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 $Q_r = \nabla \cdot q_r$.

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:

$$\text{upside: rflux_u} = \epsilon_u \sigma (T_{\text{amb, u}}^4 - T^4)$$

$$\text{downside: rflux_d} = \epsilon_d \sigma (T_{\text{amb, d}}^4 - T^4)$$

- In 1D:

$$\text{rflux_z} = \epsilon_z \sigma (T_{\text{amb, z}}^4 - T^4)$$

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:

$$\text{upside: q0_u} = h_u (T_{\text{ext, u}} - T)$$

$$\text{downside: q0_d} = h_d (T_{\text{ext, d}} - T)$$

- In 1D:

$$q0_z = h_z(T_{\text{ext},z} - T)$$

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:

$$\text{upside: } tflux_u = q0_u + rflux_u$$

$$\text{downside: } tflux_d = q0_d + rflux_d$$

- In 1D:

$$tflux_z = q0_z + rflux_z$$

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:

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

NORMAL TOTAL ENERGY FLUX (HEAT TRANSFER INTERFACE)

The variable `nteflux` is defined as:

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

NORMAL TOTAL ENTHALPY FLUX (HEAT TRANSFER INTERFACE)

The variable `nthflux` is defined as:

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

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:

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

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:

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

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:

$$\text{ndflux_u} = \text{uflux_spatial}(T)$$

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

The variable `ndflux_d` is defined as:

$$\text{ndflux_d} = \text{dflux_spatial}(T)$$



[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:

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

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

The variable `ncflux_d` is defined as:

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

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

The variable `ntflux_u` is defined as:

$$\text{ntflux_u} = \text{ndflux_u} + \text{ncflux_u}$$

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

The variable `ntflux_d` is defined as:

$$\text{ntflux_d} = \text{ndflux_d} + \text{ncflux_d}$$

Internal Boundary Energy Fluxes

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

The variable `nteflux_u` is defined as:

$$\text{nteflux_u} = \text{up}(\text{teflux}) \cdot \mathbf{un} - \text{up}(\text{dflux}) \cdot \mathbf{un} + \text{ndflux_u}$$

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

The variable `nteflux_d` is defined as:

$$\text{nteflux_d} = \text{down}(\text{teflux}) \cdot \mathbf{dn} - \text{down}(\text{dflux}) \cdot \mathbf{dn} + \text{ndflux_d}$$

INTERNAL NORMAL ENTHALPY FLUX, UPSIDE (HEAT TRANSFER INTERFACE)

The variable `nthflux_u` is defined as:

$$\text{nthflux_u} = \text{up}(\text{thflux}) \cdot \mathbf{un}$$

INTERNAL NORMAL ENTHALPY FLUX, DOWNSIDE (HEAT TRANSFER INTERFACE)

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, $Q_{\text{b,tot}}$ (SI unit: W/m^2). This variable $Q_{\text{b,tot}}$ 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 $Q_{\text{1,tot}}$ (SI unit: W/m).

The sum of the point heat sources is available in a variable called $Q_{\text{p,tot}}$ (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:

$$p_{\text{sat}}(T_{\text{dp}}) = p_v = \text{phi} \cdot \text{psat}$$

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:

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

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

$$\gamma_{T_{\text{eq}}} = \frac{M_a p C_{p,a}}{M_v L_v}$$

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](#).

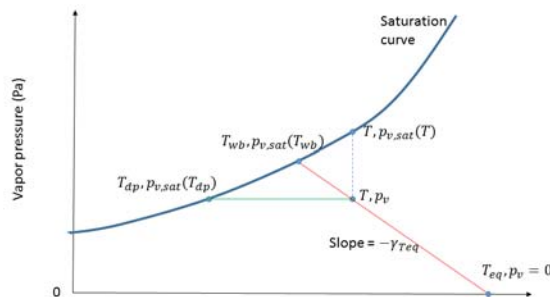


Figure 3-1: Relation between dry bulb, dew point, equivalent, and wet bulb temperatures.

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:

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

The psychrometer constant is again evaluated at temperature T .

SATURATION PRESSURE

The variable `psat` is defined by:

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

where T is the temperature. See [Functions](#) for the definition of the function `fpsat`.

RELATIVE HUMIDITY

The variable `phi` is defined by:

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

LATENT HEAT OF EVAPORATION

The variable `Lv` is defined by:

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

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
<code>phi</code>	Relative humidity	Domains
<code>cv</code>	Vapor concentration	Domains
<code>cw</code>	Total moisture concentration	Domains
<code>cl</code>	Liquid water concentration	Domains
<code>csat</code>	Saturation concentration	Domains
<code>psat</code>	Saturation pressure of water vapor	Domains
<code>pv</code>	Partial pressure of water vapor	Domains
<code>wc_l</code>	Liquid water content	Domains
<code>wc_v</code>	Vapor content	Domains
<code>omega_v</code>	Vapor mass fraction	Domains
<code>omega_moist</code>	Moisture mass fraction	Domains
<code>xvap</code>	Moisture content	Domains
<code>Lv</code>	Latent heat of evaporation	Domains
<code>Xa</code>	Molar fraction of dry air	Domains
<code>Xv</code>	Molar fraction of water vapor	Domains
<code>rhoa</code>	Density of dry air	Domains
<code>rhov</code>	Density of water vapor	Domains
<code>rho_moist</code>	Density of moist air	Domains
<code>mua</code>	Viscosity of dry air	Domains
<code>muv</code>	Viscosity of water vapor	Domains
<code>mu_moist</code>	Viscosity of moist air	Domains
<code>cflux</code>	Convective moisture flux	Domains
<code>dflux</code>	Diffusive moisture flux	Domains

TABLE 3-5: MOISTURE TRANSPORT PREDEFINED VARIABLES

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
tflux	Total moisture flux	Domains
tfluxMag	Total moisture flux magnitude	Domains
Gtot	Total moisture source	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
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
gtot	Total moisture flux	Boundaries

Moist Air Properties

PARTIAL AND SATURATION PRESSURES OF WATER VAPOR

The partial pressure of vapor p_v is the product of the saturation pressure p_{sat} and the relative humidity ϕ :

$$p_v = \phi \cdot p_{sat}$$

SATURATION CONCENTRATION OF WATER VAPOR

The saturation concentration c_{sat} is obtained from the saturation pressure p_{sat} and temperature T by:

$$c_{sat} = \frac{p_{sat}}{RT}$$

where R is the universal gas constant.

MOISTURE CONTENT

The moisture content x_{vap} is obtained from the vapor concentration c_v , the absolute pressure p_A and temperature T by:

$$x_{\text{vap}} = \frac{c_v}{\left(\frac{p_A}{RT} - c_v\right)} \cdot \frac{M_v}{M_a} = \frac{c_v}{c_a} \cdot \frac{M_v}{M_a}$$

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

VAPOR MASS FRACTION

The vapor mass fraction ω_v is obtained from the moisture content by:

$$\omega_v = \frac{c_v M_v}{\left(\frac{p_A}{RT} - c_v\right) M_a + c_v M_v} = \frac{c_v M_v}{c_a M_a + c_v M_v}$$

LIQUID, VAPOR, AND MOISTURE CONCENTRATIONS

The total moisture concentration c_w is the sum of the liquid water and vapor concentrations c_l and c_v :

$$c_w = c_v + c_l$$

Note that $c_w=0$ in moist air domains.

CONCENTRATIONS AND CONTENTS

The liquid water and vapor contents wc_l and wc_v are obtained by multiplying the concentrations c_l and c_v by the molar mass of water M_v :

$$wc_l = M_v \cdot c_l$$

$$wc_v = M_v \cdot c_v$$

LATENT HEAT OF EVAPORATION

The variable L_v is defined by:

$$L_v = l_v(T)$$

where T is the temperature. See [Functions](#) for the definition of the function L_v .

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:

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

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:

$$tflux = cflux + dflux$$

TOTAL MOISTURE FLUX MAGNITUDE

The variable `tfluxMag` is defined as:

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

Boundary Moisture Fluxes

NORMAL CONVECTIVE MOISTURE FLUX

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

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

INTERNAL NORMAL CONVECTIVE MOISTURE FLUX, UPSIDE

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

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

INTERNAL NORMAL CONVECTIVE MOISTURE FLUX, DOWNSIDE

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

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

NORMAL DIFFUSIVE MOISTURE FLUX

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

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

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:

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

INTERNAL NORMAL TOTAL MOISTURE FLUX, DOWNSIDE

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

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

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`.

Using the Boundary Conditions for the Heat Transfer Interfaces

In this section:

- [Temperature and Heat Flux Boundary Conditions](#)
- [Overriding Mechanism for Heat Transfer Boundary Conditions](#)

Temperature and Heat Flux Boundary Conditions

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

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

while the latter specifies the inward heat flux

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

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 $q_0 = h \cdot (T_{\text{ext}} - T)$, where T_{ext} is the temperature far away from the modeled domain and the heat transfer coefficient, h , represents all the physics occurring between the boundary and “far away.” It can include almost anything, but the most common situation is that h

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



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

Overriding Mechanism for Heat Transfer Boundary Conditions

Many boundary conditions are available in heat transfer. Some of these can coexist (for example, [Heat Flux](#) and [Thin Layer \(Heat Transfer Interface\)](#) 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

AIB	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



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.



Continuous Casting: Application Library path
Heat_Transfer_Module/Thermal_Processing/continuous_casting

STREAMLINE DIFFUSION

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

CROSSWIND DIFFUSION

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

Inconsistent Stabilization

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

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



In the *COMSOL Multiphysics Reference Manual*:

- [Stabilization Techniques](#)
 - [Override and Contribution](#)
-

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- \$\epsilon\$ Interface](#), and [The Turbulent Flow, Low Re k- \$\epsilon\$ 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 is 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 two-way 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 two-way 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](#)

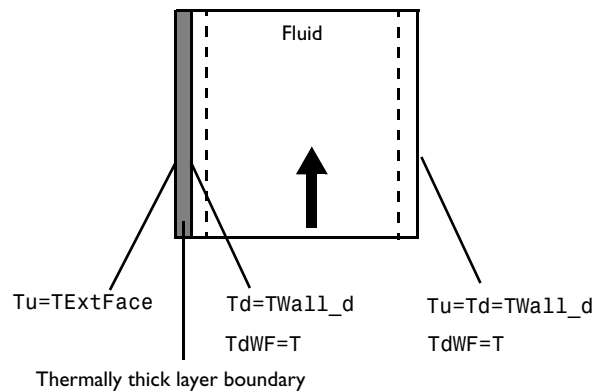
Depending on the turbulence model selected for the flow, wall functions are used or not:

- No turbulence model: no wall functions, and $Tu_{WF} = Tu$, $Td_{WF} = Td$.
- Turbulence models: wall functions detected by the Single Phase Flow physics interface. Tu_{WF} and Td_{WF} have different definitions and a flux q_{wf} 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

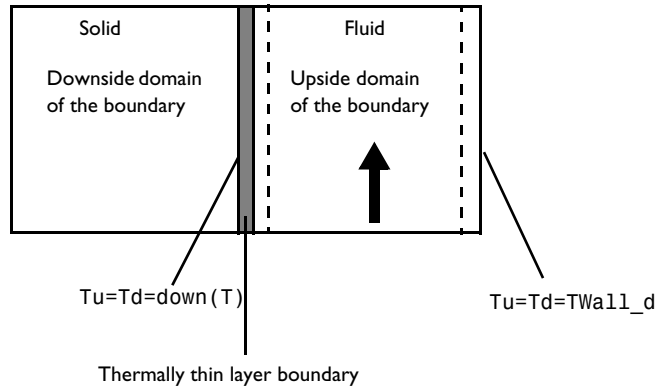
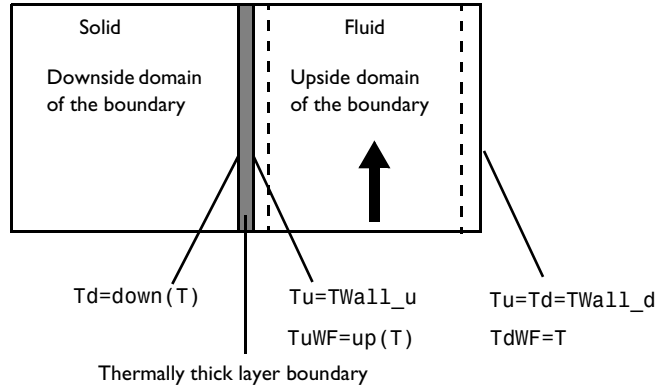
The following figure shows a configuration with a fluid domain, a thermally thick boundary condition on the left boundary, and a **Wall** feature on the right boundary. This example uses wall functions.



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

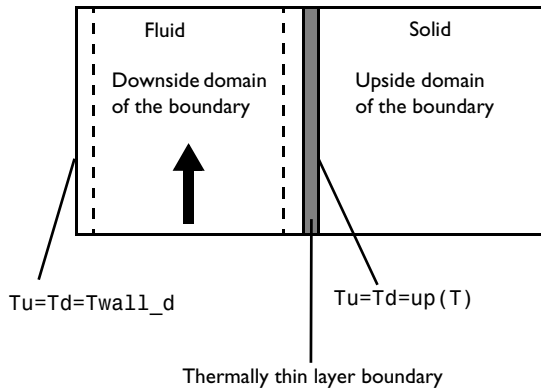
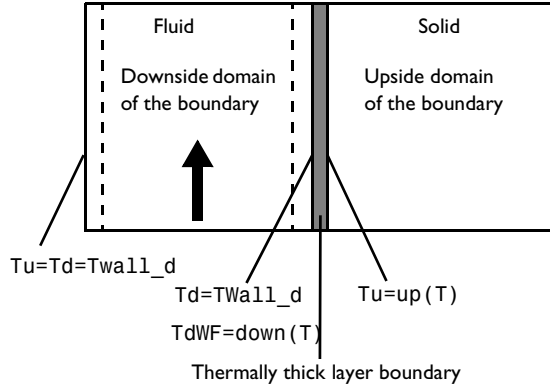
Solid in the Domain Downside the Wall

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.



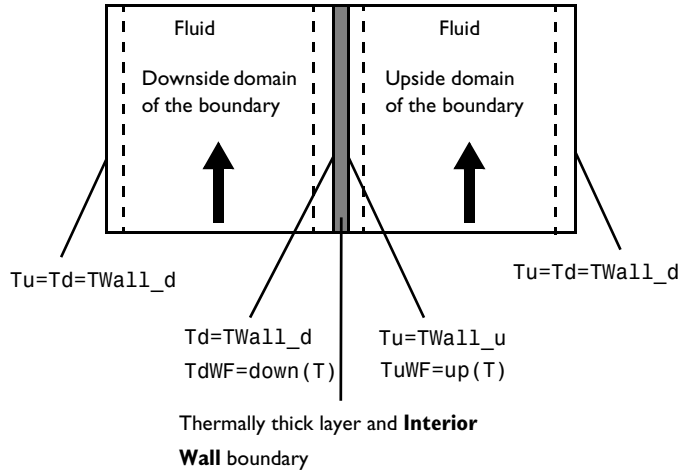
Solid in the Domain Upside the Wall

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.



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 $-\mathbf{n} \cdot \mathbf{q} = h(T_{\text{ext}} - T)$ keeps the linearity of the problem when the heat transfer coefficient, h , is constant.
 - A radiative condition of type $-\mathbf{n} \cdot \mathbf{q} = \epsilon\sigma(T_{\text{amb}}^4 - T^4)$ 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 presmoothen and postsmoothen. 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 presmoothing and postsmoothing 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, `c1_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, `c1_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 an 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, α_{necr} , 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 met (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 presmoothen and postsmoothen 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 suit 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 forces a strong coupling between temperature and radiosity variables solved by a Heat Transfer interface and a Surface-to-Surface Radiation interface, respectively. The default solver suggestion ensures that the temperature and the radiosity dependent variables are solved together, either by a fully coupled nonlinear solver, or by placing them in a same segregated group when other segregated groups are generated by some other physics interfaces.

Local Thermal Nonequilibrium

The **Local Thermal Nonequilibrium** multiphysics coupling forces a strong coupling between both the solid and the fluid temperature variables involved in the porous medium model. The default solver suggestion ensures these two temperature dependent variables are solved together, either by a fully coupled nonlinear solver, or by placing them in a same segregated group when other segregated groups are generated by some other physics interfaces.

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 presmoothing and postsmoothing 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*.

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 `s1mat1_xdim`, when a single layer material (`s1mat1`) 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 `s1mat1.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=s1mat1.th` corresponds to its upside. Upside and downside settings can be visualized by plotting the global normal vector (n_x, n_y, n_z) , that always points from downside to upside. See [Tangent and Normal Variables](#) in the *COMSOL Multiphysics Reference Manual*. Note that the normal vector $(ht.n_x, ht.n_y, ht.n_z)$ 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.

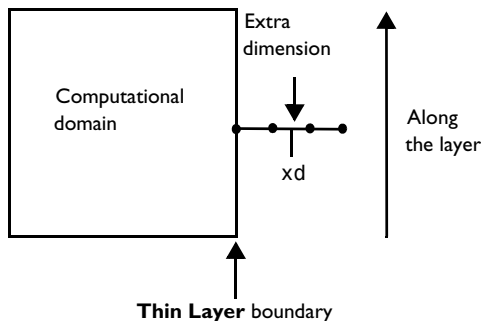


Figure 3-2: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results along the layer at the coordinate x_d .

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.

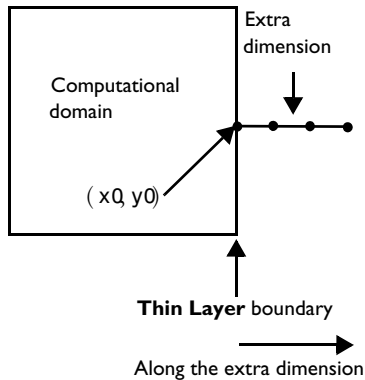


Figure 3-3: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results through the layer at the point (x_0, y_0) .

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 i^{th} layer of the layered material, 0 on the other layers.

Here, `xdimTag` is the tag of the extra dimension. For example, it may be `l1mat1_xdim` when a layered material link (`l1mat1`) 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 i^{th} layer of the layered material.



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

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 $I_{\text{sn,amb}}$ and $I_{\text{sh,amb}}$.



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** $P_{0,\text{amb}}$, the **Clear sky noon beam normal irradiance** $I_{\text{sn,amb}}$, and the **Clear sky noon diffuse horizontal irradiance** $I_{\text{sh,amb}}$ 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.



A single country may be available for more than one region selection if it has stations spread over different regions. For example, United States of America is available in the **Country** list when either North America, Eurasia, or Oceania is selected in the **Region** list.



When a station is selected **From list**, its World Meteorological Organization (WMO) reference is displayed 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.



See [Ambient Data Interpolation](#) for more information about the interpolation of data over months and hours.

See [Processing of ASHRAE Data](#) for more information about the data.



A time unit suitable for simulations over a day or a year may be set in the **Study Settings** section of the **Time Dependent** node, by using for example **h** for hour, **d** for day, or **a** for a year. See [Using Units](#) in the *COMSOL Multiphysics Reference Manual* for more details.

Ambient Conditions

Based on the measured data, several conditions are available for the **Temperature**, the **Dew point temperature**, 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	$T_{\text{amb}} = \langle T_{\text{station}} \rangle$
Low	$T_{\text{amb}} = \langle T_{\text{station}} \rangle - \sigma_{T, \text{station}}$
High	$T_{\text{amb}} = \langle T_{\text{station}} \rangle + \sigma_{T, \text{station}}$
Lowest	$T_{\text{amb}} = \min(T_{\text{station}})$
Highest	$T_{\text{amb}} = \max(T_{\text{station}})$
User defined coefficient for deviation	$T_{\text{amb}} = \langle T_{\text{station}} \rangle + c_{\sigma} \cdot \sigma_{T, \text{station}}$
User defined correction	$T_{\text{amb}} = \langle T_{\text{station}} \rangle + \Delta T$
Heating wind correlation ⁽¹⁾	$T_{\text{amb}} = \langle T_{\text{station}} \rangle + \Delta T_{\text{wind}}$
Cooling wind correlation ⁽¹⁾	$T_{\text{amb}} = \langle T_{\text{station}} \rangle - \Delta T_{\text{wind}}$

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

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.

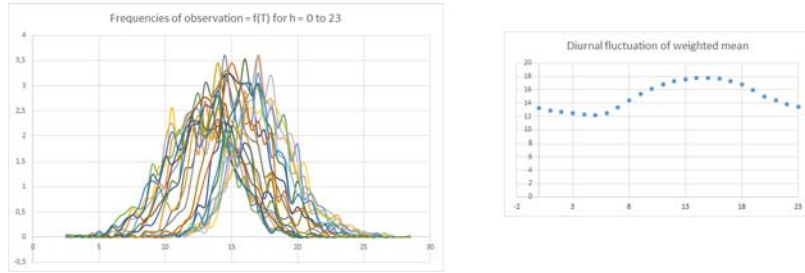


Figure 3-4: Computation of weighted mean from frequencies of observations for the diurnal fluctuations of temperature.

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:

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

- Low:

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

- High:

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

- Lowest:

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

- Highest:

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

- User defined coefficient for deviation:

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

- User defined correction:

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

where:

- $\langle T_{\text{station}} \rangle$ (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 $\langle T_{\text{station}} \rangle$.

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

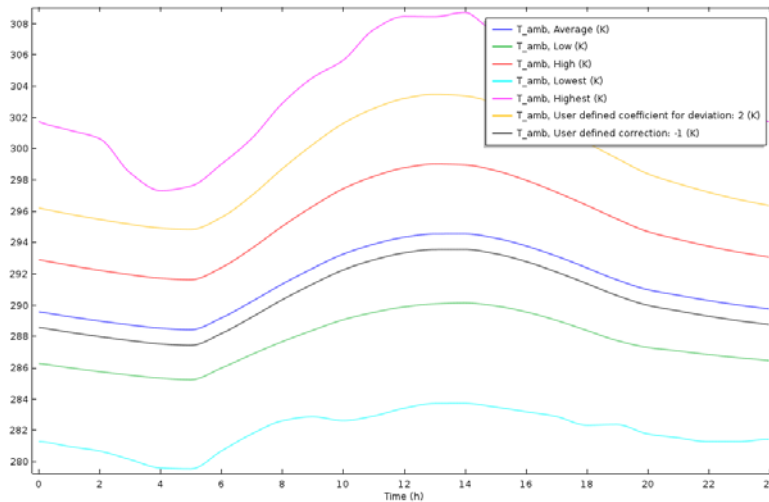


Figure 3-5: Comparison of ambient conditions for the temperature at New York/John F. Ke, on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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 $\langle T_{\text{station}} \rangle$, 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 1st of June.

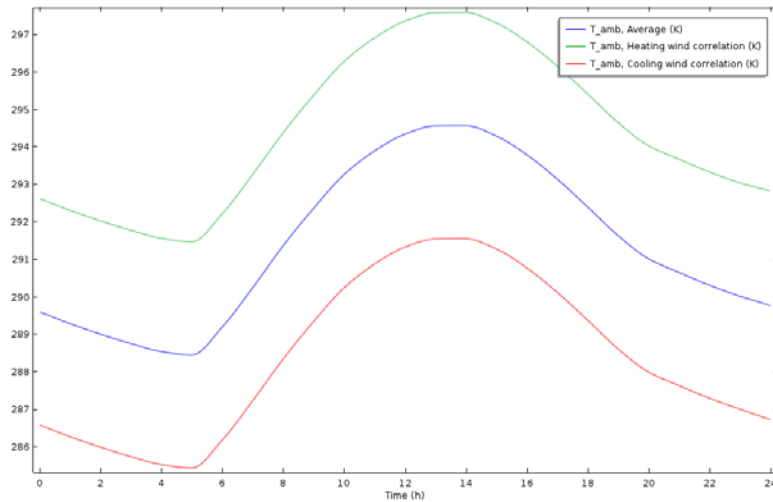


Figure 3-6: Comparison of heating and cooling wind correlations for the temperature at New York/John F. Ke, on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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:

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

- Lowest:

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

- Highest:

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

where:

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

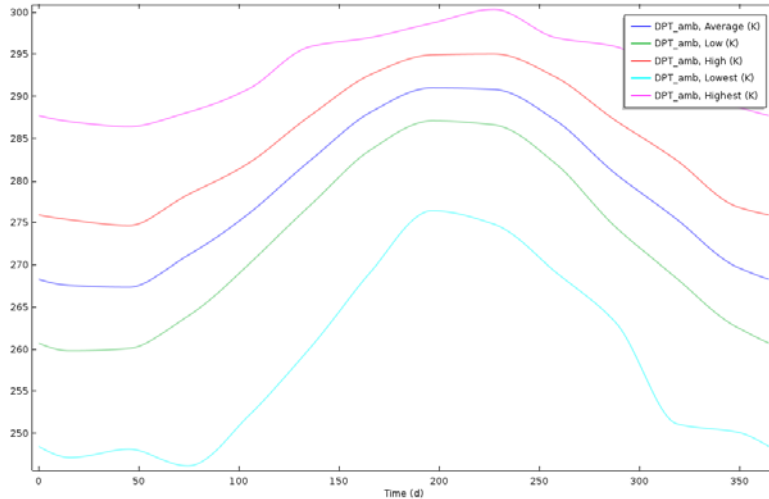


Figure 3-7: Comparison of the ambient conditions for the dew point temperature at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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.

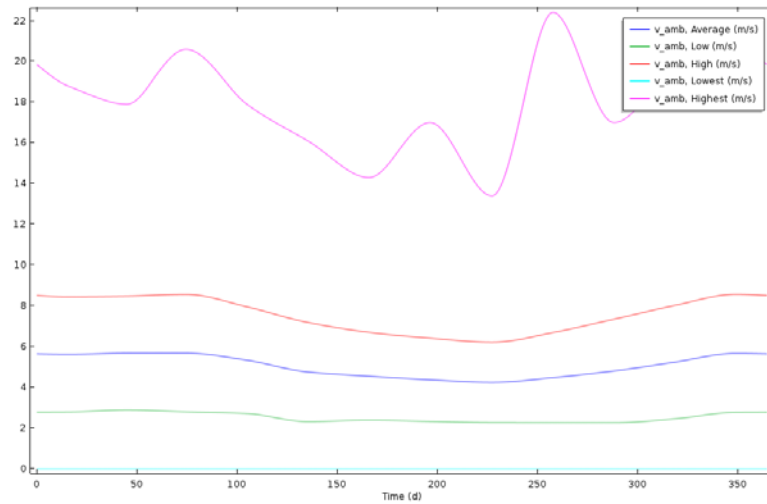


Figure 3-8: Comparison of the ambient conditions for the wind speed at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

CONDITIONS OF PRECIPITATION RATE

- Average:

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

- 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:

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

where:

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

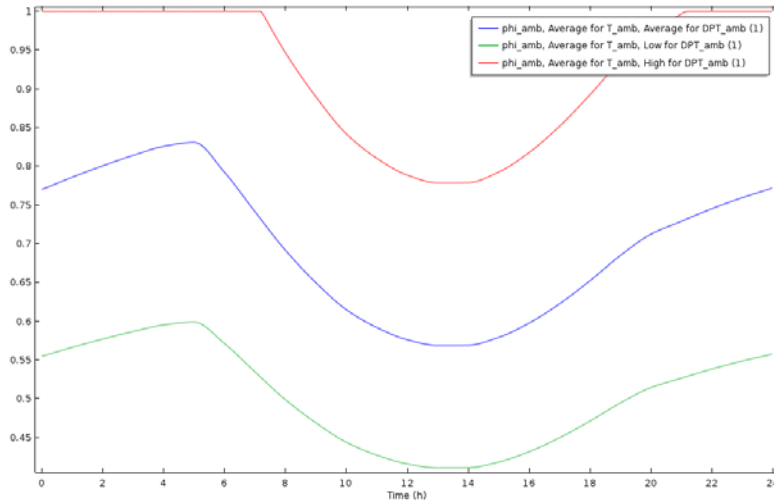


Figure 3-9: Diurnal fluctuations of relative humidity for different ambient conditions at New York/John F. Ke, on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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

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

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

See [Moisture Content](#) for more details.

DIRECT AND DIFFUSE SOLAR IRRADIANCES

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^2), defined by:

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

where $I_{sn, station}$ (SI unit: W/m^2) and $I_{sh, station}$ (SI unit: W/m^2) 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.

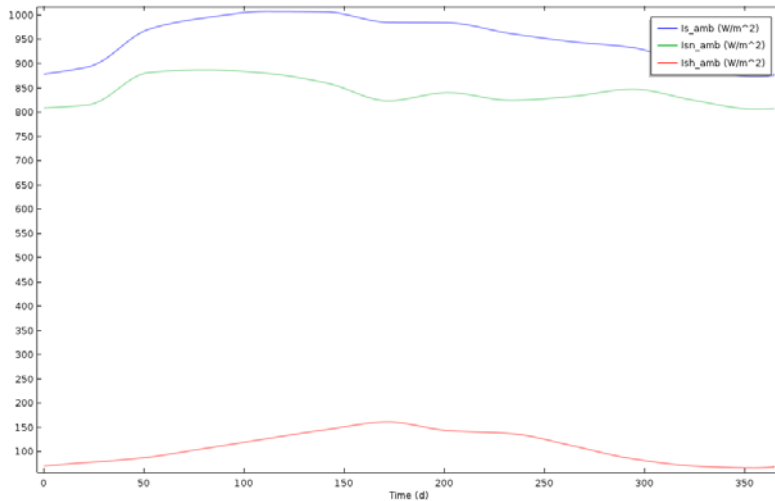


Figure 3-10: Decomposition of solar irradiance into normal and horizontal irradiance at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.)

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:

Heat Flux

General inward heat flux

Convective heat flux

$q_0 = h \cdot (T_{\text{ext}} - T)$

Heat transfer coefficient:

External forced convection

Plate, averaged transfer coefficient

Plate length:

L 1 m

Velocity, fluid:

U Wind velocity (ampr1)

Fluid:

Air

Absolute pressure:

p_A Ambient absolute pressure (ampr1)

External temperature:

T_{ext} Ambient temperature (ampr1)

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

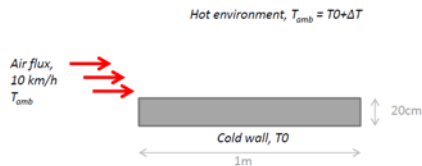
Modeling Heat Transfer by Radiation

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).

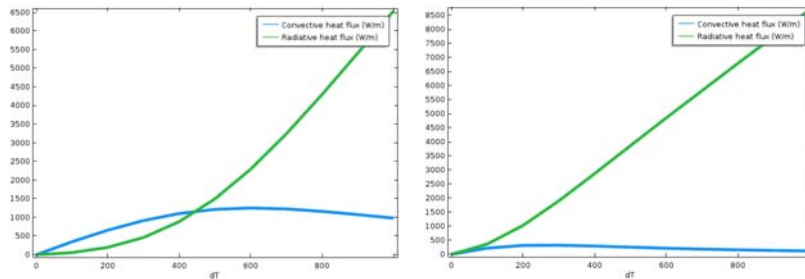


Figure 3-12: Integral of convective and radiative heat fluxes (W/m) along the top boundary, for two values of emissivity, as a function of the temperature difference.

Whereas the boundary radiative and convective heat fluxes are of the same order for temperature difference up to 500K when the emissivity is low ($\epsilon=0.1$), radiation becomes the dominant mode of heat transfer even for a small temperature difference when the emissivity is high ($\epsilon=0.9$). 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 `rf lux` 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, \dots, J_n (if multiple wavelengths)
<p>⁽¹⁾ Only with the Heat Transfer Module</p> <p>⁽²⁾ Only in 2D and 1D components</p> <p>⁽³⁾ Only in 3D and 2D components</p> <p>⁽⁴⁾ Only in 3D, 2D axi, and 2D components</p> <p>⁽⁵⁾ Only with Rosseland approximation and PI method</p>		

Interfaces and Features for Radiation in Participating Media

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 (PI method, DOM) Linear anisotropic (PI method, DOM) Anisotropic (DOM) Henyeey-Greenstein (DOM)	Isotropic (Rosseland approximation, PI method, DOM) Linear anisotropic (PI method, DOM) Anisotropic (DOM) Henyeey-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 (PI method)	Opaque Surface Semitransparent Surface Incident Intensity Continuity of Interior Boundary (DOM) Periodic Condition Symmetry (PI 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 (PI method) High (DOM)	Low (Rosseland approximation) Medium (PI 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 (PI method) Radiative intensity $I=I_1, \dots, I_n$, with n up to 512 (DOM)	None (Rosseland approximation) Incident radiation G (PI method) Radiative intensity $I=I_1, \dots, I_n$, 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 PI 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.

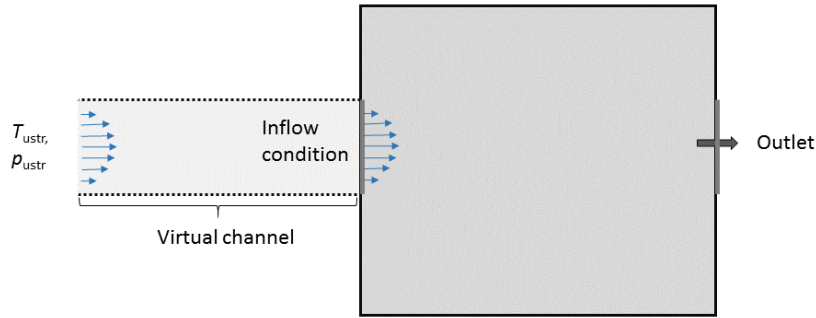


Figure 3-13: Representation of the virtual domain corresponding to an Inflow boundary condition.

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:

$$\Delta H = \int_{T_{ustr}}^{T'} C_p dT + \int_{p_{ustr}}^p \frac{1}{\rho} (1 - \alpha_p T) dp = \Delta H_T + \Delta H_p \quad (3-1)$$

where:

$$\Delta H_T = \int_{T_{ustr}}^{T'} C_p dT \quad (3-2)$$

and

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

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:

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

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.



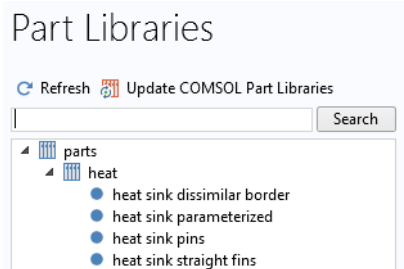
Part Libraries in the *COMSOL Multiphysics Reference Manual*

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](#).

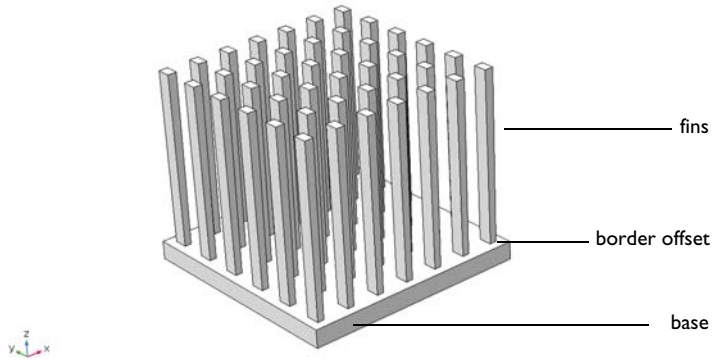
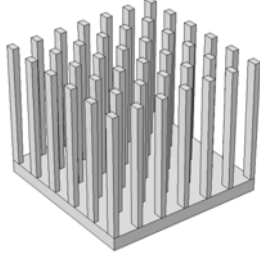
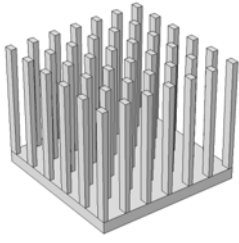
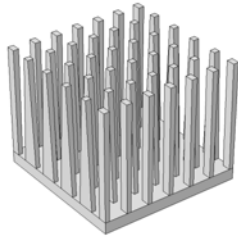
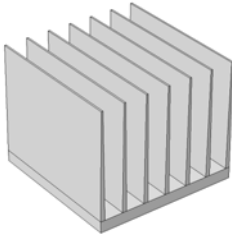


Figure 3-14: Heat sinks components

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.

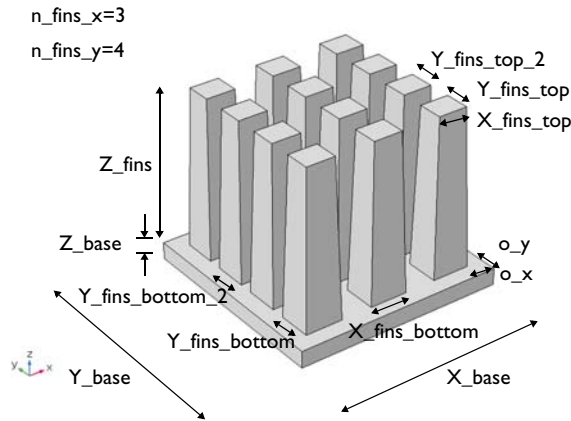


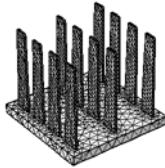
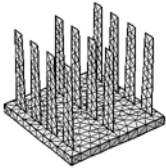
Figure 3-15: Fins and base parameters in Heat Sink — Parameterized part

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
	
<p>Mesh statistics:</p> <p>8003 tetrahedra</p> <p>5900 triangles</p> <p>1220 edge elements</p> <p>104 vertex elements</p>	<p>Mesh statistics:</p> <p>906 tetrahedra</p> <p>1058 triangles</p> <p>312 edge elements</p> <p>56 vertex elements</p>

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`).

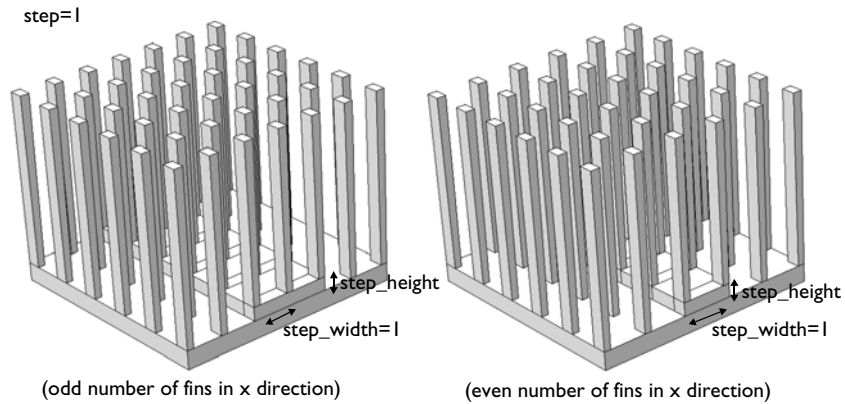


Figure 3-16: Step parameters in Heat Sink — Parameterized part

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 (`she11=1`).

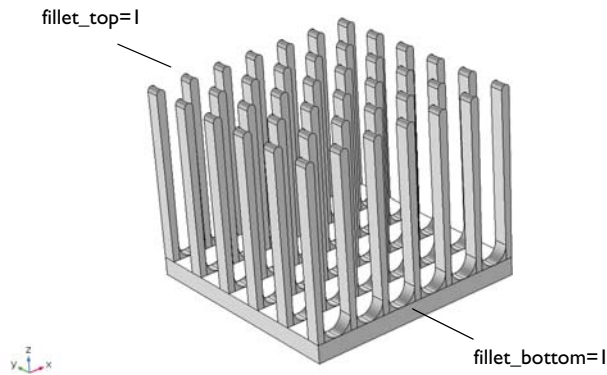


Figure 3-17: Fillet parameters in Heat Sink — Parameterized part

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](#).

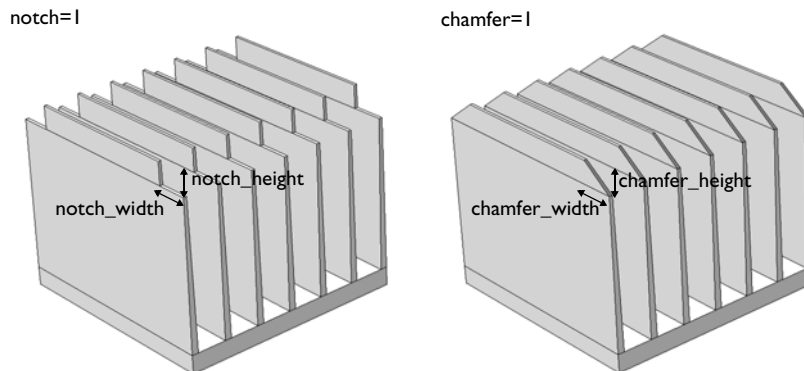



Figure 3-18: Notch and chamfer parameters in Heat Sink — Straight Fins part

References

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

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 in Moist 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
- The Heat Transfer Coefficients
- 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:

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

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:

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

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:

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

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

Here, the counterpart of the fluid pressure is the first Piola-Kirchhoff stress tensor, \mathbf{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

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

and for a solid, the definitions become:

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

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:

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

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

SENSIBLE ENTHALPY

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

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

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

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

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

$$H = H_{\text{ref}} + \int_{T_{\text{ref}}}^T C_p \cdot dT \quad (4-6)$$

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:

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

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:

$$\mathbf{q} = -k \nabla T \quad (4-8)$$

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):

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

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):

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

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):

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

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](#)):

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

where $\boldsymbol{\sigma}$ 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:

$$\boldsymbol{\sigma} = -p\mathbf{I} + \boldsymbol{\tau} \quad (4-12)$$

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

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

Equivalently, the stress power can also be expressed as:

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

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:

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

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:

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

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

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

or equivalently in the *spatial* frame:

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

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	$\rho \frac{d\mathbf{u}}{dt} = \nabla \cdot \boldsymbol{\sigma} + \mathbf{F}_v$	$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot \boldsymbol{\sigma} + \mathbf{F}_v$
Conservation of Angular Momentum	$\boldsymbol{\sigma}^T = \boldsymbol{\sigma}$	$\boldsymbol{\sigma}^T = \boldsymbol{\sigma}$

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](#):

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

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: $\text{J}/(\text{kg}\cdot\text{K})$)
- 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: $1/\text{K}$)
- 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:

$$Q_{\text{ted}} = -\alpha T \frac{dS}{dt} \quad (4-17)$$

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):

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

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/m³)
- 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/m²)
- \mathbf{q}_r is the heat flux by radiation (SI unit: W/m²)
- 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/m³)

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.

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

The second term represents viscous dissipation in the fluid:

$$Q_{vd} = \boldsymbol{\tau} : \nabla \mathbf{u} \quad (4-20)$$

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:

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

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:

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

where T_{upstream} is the upstream temperature, T is the inlet temperature, p_{upstream} 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:

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

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:

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

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/(m²·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 :

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

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. Without condensation, the moisture content is not affected by temperature and pressure. The moisture content represents a ratio of mass, and it is thus a dimensionless number.

RELATIVE HUMIDITY

The *relative humidity* of an air mixture is expressed as follows:

$$\phi_w = \frac{p_v}{p_{\text{sat}}} \quad (4-25)$$

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, $m_{\text{tot}} = m_v + m_a$:

$$\omega = \frac{m_v}{m_{\text{tot}}} \quad (4-26)$$



When the water vapor only accounts for a few percent in the total mass, the moisture content and the specific humidity are very close: $x_{\text{vap}} \approx \omega$ (only for low values). For larger values of ω , the two quantities are more precisely related by:

$$x_{\text{vap}} = \frac{\omega}{1 - \omega}$$

CONCENTRATION

The concentration is defined by:

$$c_v = \frac{n_v}{V} \quad (4-27)$$

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:

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

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

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

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:

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

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

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:

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

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:

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

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

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

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. 36 and Ref. 37, the dynamic viscosity is defined as:

$$\mu_m = \sum_{i=a,v} \frac{X_i \mu_i}{\sum_j X_j \phi_{ij}} \quad (4-35)$$

where ϕ_{ij} is given by

$$\phi_{ij} = \frac{\left[1 + \left(\frac{\mu_i}{\mu_j} \right)^{\frac{1}{2}} \left(\frac{M_j}{M_i} \right)^{\frac{1}{4}} \right]^2}{\left[8 \left(1 + \frac{M_i}{M_j} \right) \right]^{\frac{1}{2}}}$$

Here, μ_a and μ_v are the dynamic viscosity of dry air and steam, respectively.

Thermal Conductivity

According to Ref. 37 and Ref. 36, the thermal conductivity of the mixture is defined similarly:

$$k_m = \sum_{i=a,v} \frac{X_i k_i}{\sum_j X_j \phi_{ij}} \quad (4-36)$$

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. 38 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 x_{vap} .
- Vapor mass fraction ω_{moist} .
- Concentration of water vapor c .
- Relative humidity ϕ_i . 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:

- $\text{ht.feature.fc}(\text{RH}, T, pA)$, where RH is the relative humidity $0 \leq \phi_w \leq 1$, 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:

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



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

- $\text{ht.feature.fxvap}(\text{RH}, T, pA)$, where RH is the relative humidity $0 \leq \phi_w \leq 1$, 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. 38](#), which provides steam properties based on the Industrial Formulation IAPWS-IF97. The temperature-dependency is as shown on [Figure 4-1](#).

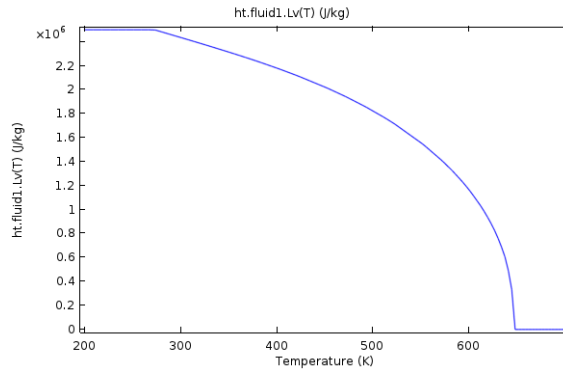


Figure 4-1: Latent heat of evaporation as a function of temperature.

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:

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

The different quantities involved here are recalled below:

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

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

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

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

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:

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

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 $A = 7.39 \cdot 10^{39} \text{ s}^{-1}$ and $\Delta E = 2.577 \cdot 10^5 \text{ J/mol}$. 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:

$$\theta_d = \min(\max(\alpha, 0), 1) \quad (4-39)$$

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, $k_{\text{eff}} = \theta_d k_d + (1 - \theta_d)k$
- The effective heat capacity at constant pressure, $(\rho C_p)_{\text{eff}} = \theta_d \rho_d C_{p,d} + (1 - \theta_d)\rho C_p$

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

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](#)). 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:

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

The mixture rule applies by multiplying the first equation by the solid volume fraction, θ_s , multiplying the second one by the porosity, ε_p , and summing resulting equations.

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

$$T_f = T_s = T \quad (4-40)$$

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

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. 31](#) 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. 32 provides criteria based on the dimensionless Sparrow number, Sp , to indicate if temperature equilibrium is still valid or if a nonequilibrium point of view should be preferred. In Ref. 33, the influence of the Darcy number, Da , and the ratio of phase conductivities is examined for transient heat transfer in packed beds. The Sparrow and Darcy numbers are defined by:

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

where:

- h_{sf} is the interstitial heat transfer coefficient between solid and fluid phases (SI unit: $W/(m^2 \cdot K)$)
- L is the plate layer thickness (SI unit: m)
- k_{eff} is the equivalent thermal conductivity of the porous medium (SI unit: $W/(m \cdot K)$)
- 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. 32 and Ref. 33, small values of Sp (less than 100 or 500) and large values of Da (from order of magnitude 10^{-7}) indicate discrepancies of temperature in each phase. However, in general, assessing the validity of local thermal equilibrium assumption remains not straightforward in specific situations. The **Local Thermal 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 frame for heat transfer in porous media where evaluating the validity of the equilibrium hypothesis is not required anymore.

Local Thermal Equilibrium

The local thermal equilibrium hypothesis of Equation 4-40 implies a common temperature, T , for both solid and fluid phase. **The Heat Transfer in Porous Media Interface** solves for the following version of the heat equation (Ref. 16), reformulated using T :

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

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

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](#), three models are proposed for an isotropic medium:

- If the heat conduction occurs in parallel in the solid and the fluid, then the effective thermal conductivity is the weighted arithmetic mean of the conductivities k_f and k_s :

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

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 :

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

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

- A last estimate is given by the weighted geometric mean of k_f and k_s :

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

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

When k_f and k_s are equal the three models give the same effective thermal conductivity.

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](#)):

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

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

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

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

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

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

$$\mathbf{u}_f = \frac{\mathbf{u}_p}{\varepsilon_p}$$

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

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

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

The [Local Thermal Nonequilibrium](#) multiphysics coupling adds the exchanged opposite heat sources $q_{sf}(T_f - T_s)$ and $q_{sf}(T_s - T_f)$ that one phase receives from or releases to the other when respective temperatures differ. The porous temperature, T , has the following definition ([Ref. 34](#)):

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

CORRELATION FOR THE INTERSTITIAL CONVECTIVE HEAT TRANSFER COEFFICIENT

The [Local Thermal Nonequilibrium](#) multiphysics feature provides a built-in correlation for q_{sf} in the spherical pellet bed configuration (2.14, 2.15, and 2.16 in [Ref. 13](#)):

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

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

$$a_{sf} = \frac{6\theta_s}{2r_p}$$

The interstitial heat transfer coefficient, h_{sf} (SI unit: W/(m²·K)), satisfies the relation:

$$\frac{1}{h_{sf}} = \frac{2r_p}{k_f \text{Nu}} + \frac{2r_p}{\beta k_s}$$

where $\beta = 10$ for spherical particles, and Nu is the fluid-to-solid Nusselt number derived from following correlation ([Ref. 14](#)):

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

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

$$\text{Pr} = \frac{\mu C_{p,f}}{k_f} \quad \text{Re}_p = \frac{2r_p \rho_f \|\mathbf{u}_p\|}{\mu}$$

VOLUMETRIC AND SURFACE THERMAL CONDITIONS

Because the Local Thermal Nonequilibrium multiphysics coupling multiplies each energy equation by its volume fraction, θ_s and ε_p for solid and fluid phases respectively, a heat source or heat flux defined in a couple heat transfer interface is also accounted with that ratio. As shown in [Equation 4-43](#) and [Equation 4-44](#), the volumetric heat sources $\theta_s Q_s$ and $\varepsilon_p Q_f$ are applied to the energy equations while the [Heat Source](#) features of each physics interface specify Q_s and Q_f .

Theory for Heat Transfer 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:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + (\rho_g C_{p,g} \mathbf{u}_g + \rho_l C_{p,l} \mathbf{u}_l) \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + Q_{\text{evap}} \quad (4-45)$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \quad (4-46)$$

The different quantities appearing here are:

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

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

- 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}_l (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: $\text{W}/(\text{m}\cdot\text{K})$) 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:

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

See [Theory for Moisture Transport in Porous Media](#) for the definition of \mathbf{g}_w and \mathbf{g}_{lc} .

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

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

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 where 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:

- [Stefan Condition](#)
- [Apparent Heat Capacity Method](#)

Stefan Condition

The *Stefan condition* defines explicitly the velocity of the phase change interface in function of the material density, the heat flux across the interface, and the latent heat of phase change.

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

The **Phase Change Interface** boundary condition should be used on the boundaries adjacent to both phases domains, to compute the time-dependent location of the phase change interface.

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

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

$$v_n = \frac{\mathbf{q}}{\rho_s L_{s \rightarrow f}}$$

with $L_{s \rightarrow f}$ the latent heat of phase change from solid to fluid (SI unit: J/kg), ρ_s the solid density (SI unit: kg/m³), and \mathbf{q} the conductive heat flux jump across the interface (SI unit: W/m²), 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.

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:

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

$$H = \frac{1}{\rho}(\theta\rho_1H_1 + (1 - \theta)\rho_2H_2)$$

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:

$$C_p = \frac{1}{\rho}(\theta_1\rho_1C_{p,1} + \theta_2\rho_2C_{p,2}) + (H_2 - H_1)\frac{d\alpha_m}{dT}$$

Here, θ_1 and θ_2 are equal to θ and $1 - \theta$, respectively. See [Figure 4-2](#) for details.

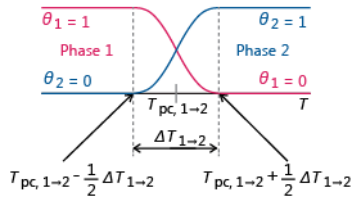


Figure 4-2: Phase indicators, phase change temperature, and transition interval.

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} :

$$C_{eq} = \frac{1}{\rho}(\theta_1\rho_1C_{p,1} + \theta_2\rho_2C_{p,2})$$

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:

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

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. 15](#):

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = Q \quad (4-47)$$

$$\mathbf{q} = -(k_{\text{eff}} \nabla T + L_v \delta_p \nabla(\phi_w p_{\text{sat}})) \quad (4-48)$$

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: $\text{J}/(\text{m}^3 \cdot \text{K})$) 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: $\text{J}/(\text{kg} \cdot \text{K})$) 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: $\text{J}/(\text{kg} \cdot \text{K})$) is the water heat capacity at constant pressure.

- k_{eff} (SI unit: $\text{W}/(\text{m} \cdot \text{K})$) 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: $\text{W}/(\text{m} \cdot \text{K})$) 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:

$$T(t) = T_0 + T^r e^{j\omega t} \quad (4-49)$$

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^r 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^r , is complex-valued since it includes the phase term $e^{j\phi}$.

From the temperature decomposition in [Equation 4-49](#), and according to the heat transfer equation in [Equation 4-16](#), heat transfer by conduction in solids is then governed by:

$$j\omega\rho C_p T^r e^{j\omega t} + \nabla \cdot (-k\nabla T_0) + \nabla \cdot (-k\nabla T^r) e^{j\omega t} = Q + Q' e^{j\omega t}$$

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:

$$j\omega\rho C_p T^r + \nabla \cdot (-k\nabla T^r) = Q'$$

which is the governing equation for T' . For constant material properties, the linearized form reads:

$$j\omega\rho_0 C_{p,0} T' + \nabla \cdot (-k_0 \nabla T') = Q' \quad (4-50)$$

and for nonlinear material properties:

$$j\omega\rho_0 C_{p,0} T' + \nabla \cdot \left(-k_0 \nabla T' - T' \left(\frac{\partial k}{\partial T} \right)_{T_0} \nabla T_0 \right) = Q' \quad (4-51)$$

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 4-50](#) or [Equation 4-51](#), 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:

$$m \overline{C_p} \frac{dT}{dt} + \int_S (\mathbf{n} \cdot \mathbf{q}) ds = \int_V Q dv \quad (4-52)$$

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

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

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

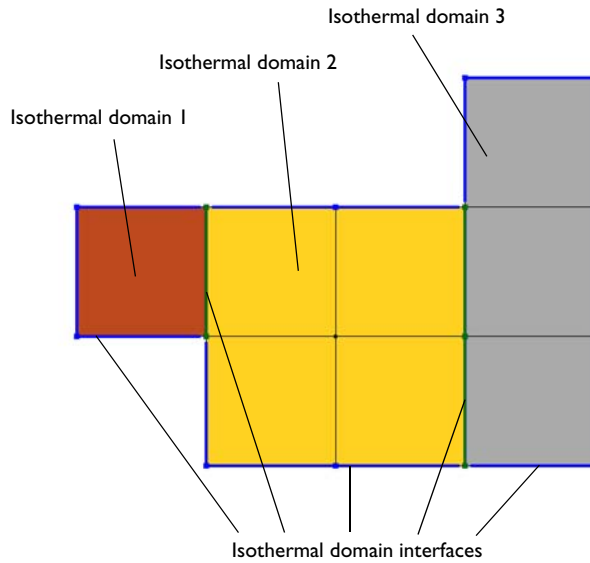


Figure 4-3: Interfaces at exterior boundaries of each Isothermal Domain.

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 $\varphi_{d \rightarrow u}$ or $\varphi_{u \rightarrow d}$, respectively, along or opposite to the geometrical normal vector. The Ventilation condition is written

$$-\mathbf{n}_d \cdot \mathbf{q}_d = \phi_{d \rightarrow u} H_d - \phi_{u \rightarrow d} H_u \quad (4-53)$$

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

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -h(T_u - T_d) \quad (4-54)$$

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

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -\frac{T_u - T_d}{R_{t,s}} \quad (4-55)$$

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.

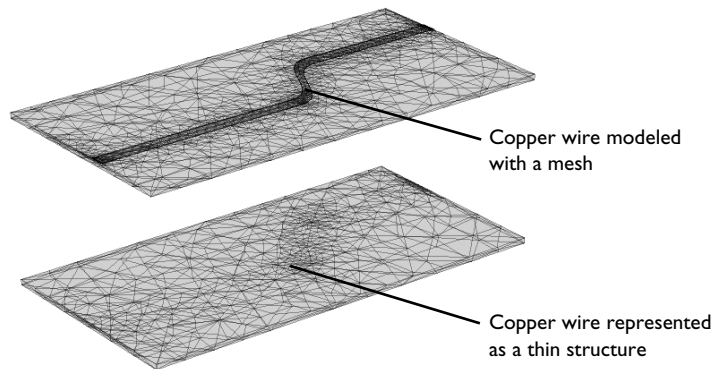


Figure 4-4: Modeling a copper wire as a domain (top) requires a denser mesh compared to modeling it as a boundary with a conductive layer (bottom).

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:

$$\nabla T = \nabla_{\mathbf{t}}T + \nabla_{\mathbf{n}}T \quad (4-56)$$

This relation simplifies to:

$$\nabla T = \nabla_{\mathbf{t}}T \quad \text{or} \quad \nabla T = \nabla_{\mathbf{n}}T$$

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 4-56 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:

$$\nabla T = \left[\left(\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} \right)^{-1} \right]^T \cdot (\nabla_{\mathbf{t}} T + \nabla_{\mathbf{n}} T)$$

with

$$\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} = \mathbf{I} + \left(z + \frac{d}{2}(-1 + z_{\text{off}}) \right) \nabla_{\mathbf{t}} \mathbf{n}$$

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 $\text{dtang}()$ operator would correspond to $\nabla_{\mathbf{n}}$ since it performs the derivation along the extra line. In the thin structure boundary, $\text{dtang}()$ 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

$$\nabla T = \left[\left(\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} \right)^{-1} \right]^T \cdot \left(\nabla_{\mathbf{t}} T + \frac{\nabla_{\mathbf{n}} T}{lsc} \right)$$

$$\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} = \mathbf{I} + \left(z + \frac{d}{2}(-1 + z_{\text{off}}) \right) \nabla_{\mathbf{t}} (lsc \mathbf{n})$$

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, $\mathbf{e}^{x, loc}$, is the surface tangent vector \mathbf{t}_1 , and the local z direction, $\mathbf{e}^{z, loc}$, is the normal vector \mathbf{n} . Their cross product defines the third orthogonal direction such that:

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

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

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

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

$$k_{bnd} = A k A^T$$

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:

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

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 4-56](#), the energy equation becomes

$$\rho \frac{\partial E_s}{\partial t} + \rho \mathbf{u} \cdot (\nabla_{\mathbf{t}} E_s + \nabla_{\mathbf{n}} E_s) + \nabla \cdot (\mathbf{q} + \mathbf{q}_r) = -(\boldsymbol{\sigma} : \mathbf{D}) + Q \quad (4-57)$$

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 4-57](#) comes along with Fourier's law of conduction:

$$\mathbf{q} = -k(\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \quad (4-58)$$

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

$$T_d = (T_s)_{L=0} \quad T_u = (T_s)_{L=d_s} \quad (4-59)$$

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.

THEMALLY THIN APPROXIMATION

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

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

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

With these assumptions, Equation 4-15 becomes:

$$d_s \rho \frac{\partial E}{\partial t} + d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{t}} E + \nabla_{\mathbf{t}} \cdot (\mathbf{q}_s + \mathbf{q}_r) = -d_s (\boldsymbol{\sigma} : \mathbf{D}) + d_s Q + q_0 \quad (4-60)$$

$$\mathbf{q}_s = -d_s k \nabla_{\mathbf{t}} T \quad (4-61)$$

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 4-60 and Equation 4-61 have an additional factor, d_z , to account for the out-of-plane thickness.

When Equation 4-60 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:

$$-\mathbf{n} \cdot \mathbf{q} = d_s Q_s - d_s \rho \frac{\partial E}{\partial t} - (d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{t}} E) - \nabla_{\mathbf{t}} \cdot (\mathbf{q}_s + \mathbf{q}_r) \quad (4-62)$$

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:

$$d_s \rho \frac{\partial E}{\partial t} + d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{n}} E + \nabla_{\mathbf{n}} \cdot (\mathbf{q}_s + \mathbf{q}_r) = -d_s (\boldsymbol{\sigma} : \mathbf{D}) + d_s Q + q_0 \quad (4-63)$$

$$\mathbf{q}_s = -d_s k \nabla_{\mathbf{n}} T \quad (4-64)$$

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 4-63](#) and [Equation 4-64](#) have an additional factor, d_z , to account for the out-of-plane thickness.

When [Equation 4-63](#) 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:

$$-\mathbf{n} \cdot \mathbf{q} = d_s Q_s - d_s \rho \frac{\partial E}{\partial t} - (d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{n}} E) - \nabla_{\mathbf{n}} \cdot (\mathbf{q}_s + \mathbf{q}_r) \quad (4-65)$$

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:

$$-k_s \frac{T_u - T_d}{d_s} \quad -k_s \frac{T_d - T_u}{d_s}$$

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:

$$-\mathbf{n}_d \cdot \mathbf{q}_d = \frac{d_s Q}{2} - \rho C_p \left(\frac{d_s}{2} \frac{\partial T_d}{\partial t} + \mathbf{u}_d \cdot \mathbf{n}_d T_d \right) - \left(-k_s \frac{T_u - T_d}{d_s} \right) - (\mathbf{q}_{r,d} \cdot \mathbf{n}_d) \quad (4-66)$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = \frac{d_s Q}{2} - \rho C_p \left(\frac{d_s}{2} \frac{\partial T_u}{\partial t} + \mathbf{u}_u \cdot \mathbf{n}_u T_u \right) - \left(-k_s \frac{T_d - T_u}{d_s} \right) - (\mathbf{q}_{r,u} \cdot \mathbf{n}_u) \quad (4-67)$$

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:

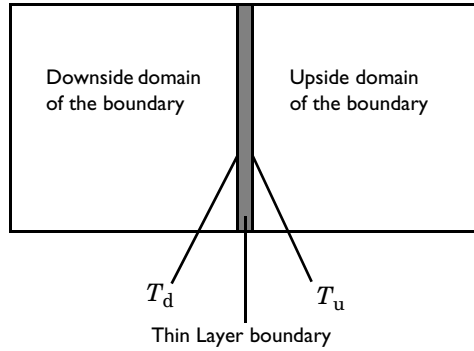


Figure 4-5: Upside and downside temperatures at a thin layer applied on an interior boundary. The thin layer is represented by the gray domain.

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:

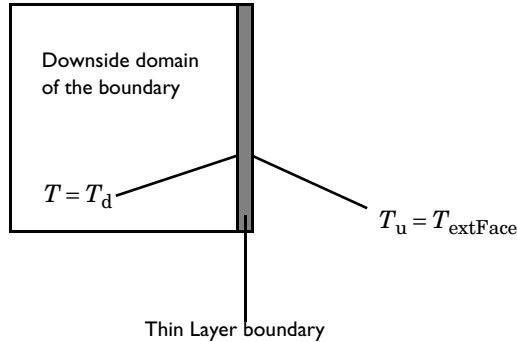


Figure 4-6: Upside and downside temperatures at a thin layer applied on an exterior boundary.

FORMULATIONS AVAILABLE WITHIN THE FEATURES

Table 4-3 summarizes the formulations available within the thin structure features of the Heat Transfer (ht) and Heat Transfer in 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 4-57 to give:

$$\rho_{si} C_{p,si} \frac{\partial T_s}{\partial t} + \nabla_{\mathbf{t}} \cdot \mathbf{q}_{si} = Q_{si} \quad (4-68)$$

$$\mathbf{q}_{si} = -k_{si} (\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \quad (4-69)$$

$$T_d = (T_s)_{L=0}$$

$$T_u = (T_s)_{L=d_s}$$

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: $\text{J}/(\text{kg}\cdot\text{K})$)
- k_{si} is the thermal conductivity of layer i (SI unit: $\text{W}/(\text{m}\cdot\text{K})$)

- 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 4-68](#) and [Equation 4-69](#).

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 4-60](#) to [Equation 4-62](#). Inside the thin layer, the heat equation becomes:

$$d_s \rho C_{p,s} \frac{\partial T}{\partial t} + \nabla_t \cdot \mathbf{q}_s = d_s Q_s + q_0 \quad (4-70)$$

$$\mathbf{q}_s = -d_s k \nabla_t T \quad (4-71)$$

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 4-70](#) and [Equation 4-71](#) 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 4-62](#), is received from the layer:

$$-\mathbf{n} \cdot \mathbf{q} = d_s \mathbf{Q}_s - d_s \rho C_{p,s} \frac{\partial T}{\partial t} - \nabla_{\mathbf{t}} \cdot \mathbf{q}_s \quad (4-72)$$



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 4-72](#). See [The Heat Transfer in Shells Interface](#) for more information about the physics interface solving [Equation 4-70](#).



- *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 4-65](#) and gives

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -\frac{1}{2} d_s \rho_s C_{p,s} \frac{\partial T_d}{\partial t} - k_s \frac{T_u - T_d}{d_s} + \frac{1}{2} d_s \mathbf{Q}_s \quad (4-73)$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = -\frac{1}{2} d_s \rho_s C_{p,s} \frac{\partial T_u}{\partial t} - k_s \frac{T_d - T_u}{d_s} + \frac{1}{2} d_s \mathbf{Q}_s \quad (4-74)$$

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 4-75](#) and [Equation 4-76](#):

$$d_{\text{tot}} = \sum_{j=1}^n d_{sj} \quad (4-75)$$

$$k_{\text{tot}} = \frac{d_{\text{tot}}}{\sum_{j=1}^n \frac{d_{sj}}{k_{sj}}} \quad (4-76)$$

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 4-73](#) and [Equation 4-74](#).

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 4-57](#) to give:

$$\rho C_p \frac{\partial T_s}{\partial t} + \rho C_p \mathbf{u} \cdot (\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) + \nabla_{\mathbf{t}} \cdot \mathbf{q}_f = Q_f \quad (4-77)$$

$$\mathbf{q}_f = -k(\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \quad (4-78)$$

$$T_d = (T_s)_{L=0}$$

$$T_u = (T_s)_{L=d_f}$$

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: $\text{J}/(\text{kg}\cdot\text{K})$)

- k is the thermal conductivity (SI unit: W/(m·K))
- Q_f is the heat source applied to the film (SI unit: W/m³)
- 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 4-77](#) and [Equation 4-78](#).

THERMALLY THIN APPROXIMATION

The thermally thin approximation is derived from [Equation 4-60](#) to [Equation 4-62](#). Inside the thin layer, the heat equation becomes:

$$d_f \rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla_{\mathbf{t}} T \right) + \nabla_{\mathbf{t}} \cdot \mathbf{q}_f = d_f Q_f + q_0 \quad (4-79)$$

$$\mathbf{q}_f = -d_f k \nabla_{\mathbf{t}} T \quad (4-80)$$

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 4-70](#) and [Equation 4-71](#) 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 4-62](#), is received from the layer:

$$-\mathbf{n} \cdot \mathbf{q} = d_f Q_f - d_f \rho C_p \frac{\partial T}{\partial t} - d_f \rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T - \nabla_{\mathbf{t}} \cdot \mathbf{q}_f \quad (4-81)$$



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 4-79](#). See [The Heat Transfer in Films Interface](#) for more information about the physics interface solving [Equation 4-81](#).

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 than 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 4-60](#) to [Equation 4-62](#) 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:

$$d_{\text{fr}}(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + d_{\text{fr}} \rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T + \nabla_{\mathbf{t}} \cdot \mathbf{q}_{\text{fr}} = d_{\text{fr}} Q + q_0 \quad (4-82)$$

$$\mathbf{q}_{\text{fr}} = -d_{\text{fr}} k_{\text{eff}} \nabla_{\mathbf{t}} T \quad (4-83)$$

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}_{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 4-62](#), is received from the fracture:

$$-\mathbf{n} \cdot \mathbf{q} = d_{\text{fr}} Q_0 - d_{\text{fr}} (\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} - d_{\text{fr}} \rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T - \nabla_{\mathbf{t}} \cdot \mathbf{q}_{\text{fr}} \quad (4-84)$$



See [Fracture \(Heat Transfer Interface\)](#) and [Porous Medium \(Heat Transfer in Shells Interface\)](#) for more information about the boundary feature solving [Equation 4-84](#). See [The Heat Transfer in Fractures Interface](#) for more information about the physics interface solving [Equation 4-82](#).

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:

$$\int_{S(R)} Q ds = A_1 Q_1 - A_1 \rho_1 C_{p,1} + \frac{\partial T}{\partial t} - \nabla_{\mathbf{t}} \cdot \mathbf{q}_1 \quad (4-85)$$

$$\mathbf{q}_1 = -A_1 k_1 \nabla_{\mathbf{t}} T \quad (4-86)$$

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 `1ts.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	$\frac{\Delta T}{R}$	$-P$	P
Convective Thermal Resistor	$\frac{\Delta T}{R_{conv}}$	$-P$	P
Radiative Thermal Resistor	$\frac{\Delta T}{R_{rad}}$	$-P$	P
Heat Pipe	$\frac{\Delta T}{R}$	$-P$	P
Expression in the other two-port components			
Thermal Capacitor	$-C \frac{\partial \Delta T}{\partial t}$	$-P$	P
Heat Rate Source	P_{src}	0	P
Thermoelectric Module	$P_{1,src} + P_{2,src}$	$P_{1,src}$	$P_{2,src}$

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

$$P = -\frac{\Delta T}{R} \quad (4-87)$$

where R (SI unit: K/W) is the thermal resistance.

The expression of the thermal resistance used in [Equation 4-87](#) 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:

$$R = \frac{1}{2\pi k H} \ln\left(\frac{r_o}{r_i}\right)$$

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 4-87 can be updated to get a relationship between the heat transfer rate P and the overall temperature difference $\Delta T_{\text{overall}}$ (SI unit: K):

$$P = -\frac{\Delta T_{\text{overall}}}{R_{\text{tot}}}$$

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, \dots, R_n , see Figure 4-7, the total resistance is defined by:

$$R_{\text{tot}} = \sum_{i=1}^n R_i$$

with

$$R_i = \frac{L_i}{k_i A}$$

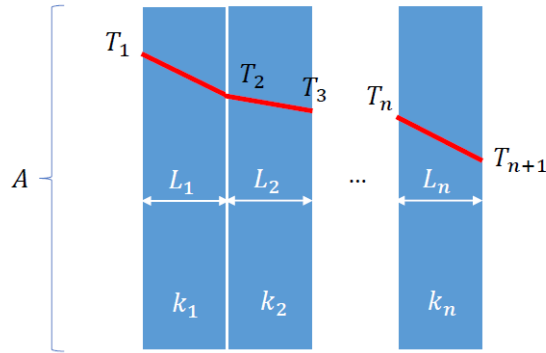
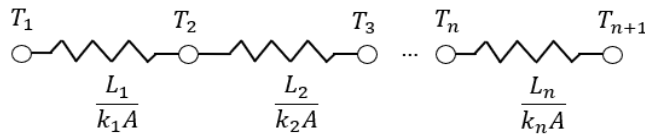


Figure 4-7: Steady conduction through a composite wall made of n layers

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, \dots, 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

$$R_i = \frac{L}{k_i A_i}$$

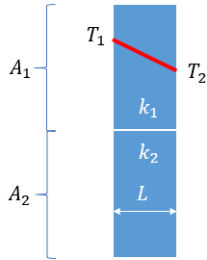
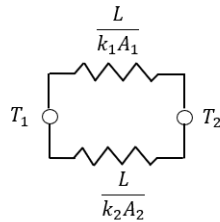


Figure 4-8: Steady conduction through a composite wall made of n slabs.

This corresponds to the following parallel thermal circuit representation:



where $\Delta T_{\text{overall}} = T_2 - T_1$. 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. 20](#) 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

$$P_{12} = \frac{A_1 \sigma (T_1^4 - T_2^4)}{\frac{1}{\epsilon_1} + \frac{1 - \epsilon_2}{\epsilon_2} \left(\frac{r_1}{r_2}\right)^2}$$

Large (infinite) Parallel Planes

The surface areas A_1 and A_2 verify

$$A_1 = A_2$$

And the net radiation exchange is

$$P_{12} = \frac{A_1 \sigma (T_1^4 - T_2^4)}{\frac{1}{\epsilon_1} + \frac{1}{\epsilon_2} - 1}$$

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

$$P_{12} = \frac{A_1 \sigma (T_1^4 - T_2^4)}{\frac{1}{\epsilon_1} + \frac{1 - \epsilon_2}{\epsilon_2} \left(\frac{r_1}{r_2}\right)}$$

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 ($\epsilon_1=\epsilon_2=1$).

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: $\text{J}/(\text{kg}\cdot\text{K})$), 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: $\text{W}/(\text{m}^2\cdot\text{K})$)
- A is the surface of convective heat transfer (SI unit: m^2)

This approximation, referred as the lumped thermal capacitance model (see [Ref. 40](#)), 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):

$$Bi = \frac{hL}{k}$$

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 $Bi < 0.1$.

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.

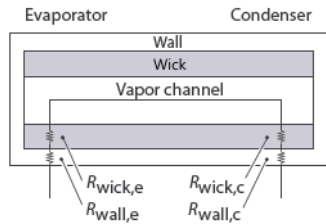


Figure 4-9: Heat pipe components.

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:

$$R = R_{\text{wall},e} + R_{\text{wick},e} + R_{\text{lv},e} + R_a + R_{\text{lv},c} + R_{\text{wick},c} + R_{\text{wall},c}$$

where

$$R_a = \frac{1}{\frac{1}{R_{v,a}} + \frac{1}{R_{\text{wick},a}} + \frac{1}{R_{\text{wall},a}}}$$

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
- $R_{v,a}$ 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:

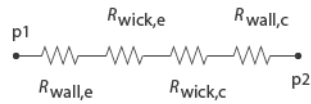


Figure 4-10: Thermal network representation of a heat pipe.

See Ref. 17 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.

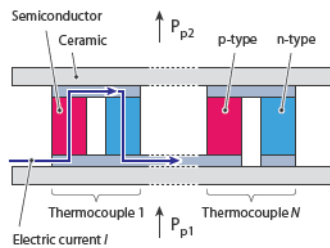


Figure 4-11: Thermoelectric module schematic description.

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:

$$P_{\text{hot}} = -SIT_{\text{hot}} + \frac{R_e I^2}{2} + \frac{\Delta T}{R} - C_{\text{hot}} \frac{\partial T_{\text{hot}}}{\partial t} \quad (4-88)$$

- Cold junction:

$$P_{\text{cold}} = SIT_{\text{cold}} + \frac{R_e I^2}{2} - \frac{\Delta T}{R} - C_{\text{cold}} \frac{\partial T_{\text{cold}}}{\partial t} \quad (4-89)$$

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 4-88](#) and [Equation 4-89](#) 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.

THEMOCOUPLER 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 4-88](#) and [Equation 4-89](#):

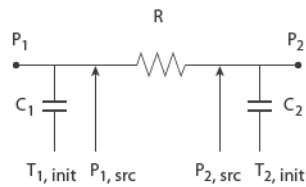


Figure 4-12: Thermal network representation of a thermoelectric module.

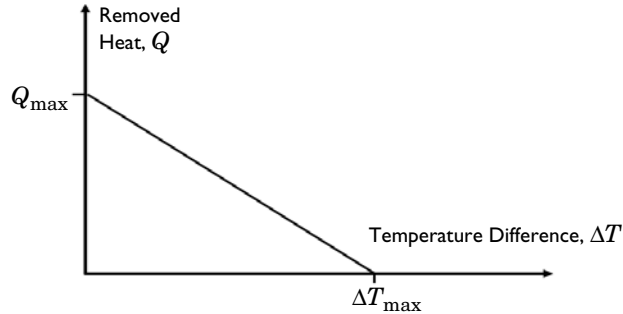
See [Ref. 17](#) and [Ref. 18](#) for details about the network representation of thermoelectric modules.

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:

$$0 \leq \Delta T \leq \Delta T_{\max}$$

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 Emissivity and Absorptivity](#)
- [The Radiosity Method for Diffuse-Gray Surfaces](#)
- [The Radiosity Method for Diffuse-Spectral Surfaces](#)
- [View Factor Evaluation](#)

Deriving the Radiative Heat Flux 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.

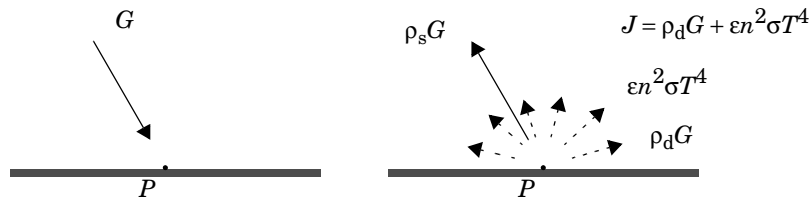


Figure 4-13: Incoming irradiation (left), outgoing radiosity (right).

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 diffusively reflected and emitted radiation:

$$J = \rho_d G + \epsilon e_b(T) \quad (4-90)$$

According to the Stefan-Boltzmann law, $e_b(T)$ is the power radiated across all wavelengths and depends on the fourth 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

$$q = (1 - \rho_s)G - J \quad (4-91)$$

Using Equation 4-90 and Equation 4-91, 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 .

$$q = (1 - (\rho_d + \rho_s))G - \epsilon e_b(T) \quad (4-92)$$

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:

$$\alpha = \epsilon = 1 - (\rho_d + \rho_s) \quad (4-93)$$

Thus, for ideal gray bodies, q is given by:

$$q = \epsilon(G - e_b(T)) \quad (4-94)$$

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.

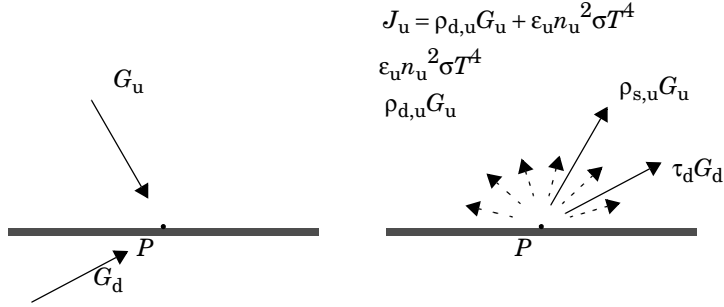


Figure 4-14: Upside and downside incoming irradiation (left), upside outgoing radiosity (right). The downside outgoing radiosity is defined in a similar way.

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:

$$J_u = \rho_{d,u}G_u + \varepsilon_u e_{b,u}(T_u) \quad (4-95)$$

$$J_d = \rho_{d,d}G_d + \varepsilon_d e_{b,d}(T_d) \quad (4-96)$$

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:

$$q_u = (1 - \rho_{s,u} - \tau_u)G_u - J_u \quad (4-97)$$

$$q_d = (1 - \rho_{s,d} - \tau_d)G_d - J_d \quad (4-98)$$

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:

$$\varepsilon_u + \rho_{d,u} = 1 - \rho_{s,u} - \tau_u \quad (4-99)$$

$$\varepsilon_d + \rho_{d,d} = 1 - \rho_{s,d} - \tau_d \quad (4-100)$$

Using Equation 4-95 to Equation 4-100, 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 :

$$q_u = \varepsilon_u(G_u - e_{b,u}(T_u)) \quad (4-101)$$

$$q_d = \varepsilon_d(G_d - e_{b,d}(T_d)) \quad (4-102)$$

Thus, for ideal gray bodies, q is given by:

$$q = \varepsilon_u(G_u - e_{b,u}(T_u)) + \varepsilon_d(G_d - e_{b,d}(T_d)) \quad (4-103)$$

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

$$\tau(\theta) = \begin{cases} \tau & \text{if } \theta \leq \theta_c \\ 0 & \text{if } \theta > \theta_c \end{cases}$$

where θ_c is the critical angle. Using the following relation

$$\rho_s(\theta) + \tau(\theta) = 1 - (\varepsilon + \rho_d)$$

we can establish

$$\rho_s(\theta) = \begin{cases} \rho_s & \text{if } \theta \leq \theta_c \\ \rho_s + \tau & \text{if } \theta > \theta_c \end{cases}$$

Wavelength Dependence of Surface Emissivity and Absorptivity

The surface properties for radiation, the emissivity, and absorptivity can be dependent on the angle of emission or absorption, the surface temperature, or the radiation wavelength. The emissivity and absorptivity are defined in Ref. 19.

The Surface-to-Surface Radiation interface in the Heat Transfer module implements the radiosity method that enables arbitrary temperature dependence and assumes that

the emissivity and absorptivity are independent of the angle of emission and absorption. It is also possible to account for wavelength dependence on the surface emissivity and absorptivity.

PLANCK SPECTRAL DISTRIBUTION

The Planck's distribution of emissive power for a blackbody in vacuum is given as a function of surface temperature and wavelength.

The blackbody hemispherical emissive power (SI unit: $W/(m^3 \cdot sr)$), is denoted $e_{b,\lambda}(\lambda, T)$, and defined as (1-37 in Ref. 19):

$$e_{b,\lambda}(\lambda, T) = \frac{2\pi n^2 C_1}{\lambda^5 \left(e^{\frac{C_2}{\lambda T}} - 1 \right)} \quad (4-104)$$

where:

- the two constants C_1 (SI unit: $W \cdot m^2 / sr$) and C_2 (SI unit: $m \cdot K$) are given by

$$C_1 = hc_0^2 \quad C_2 = \frac{hc_0}{k_B}$$

- 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).

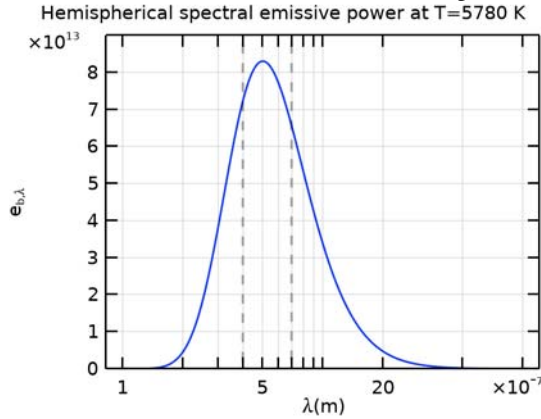


Figure 4-15: Planck distribution of a blackbody at 5780 K.

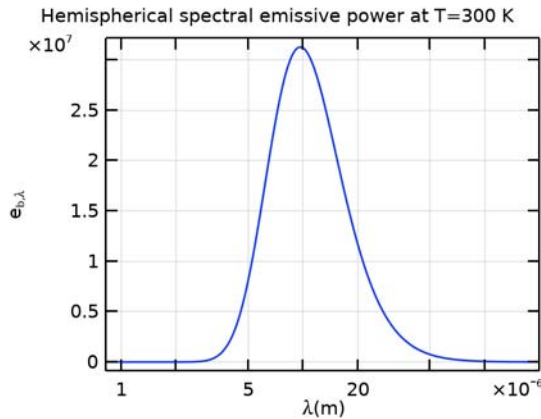


Figure 4-16: Planck distribution of a blackbody at 300 K.

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,

$$F_{\lambda_1 T \rightarrow \lambda_2 T} = \frac{\int_{\lambda_1}^{\lambda_2} e_{b,\lambda}(\lambda, T) d\lambda}{\int_0^{\infty} e_{b,\lambda}(\lambda, T) d\lambda}$$

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 $5.67 \cdot 10^{-8} \text{ W}/(\text{m}^2 \cdot \text{K}^4)$. The power radiated in the spectral band $[\lambda_1, \lambda_2]$ becomes:

$$\int_{\lambda_1}^{\lambda_2} e_{b,\lambda}(\lambda, T) d\lambda = F_{\lambda_1 T \rightarrow \lambda_2 T} e_b(T)$$

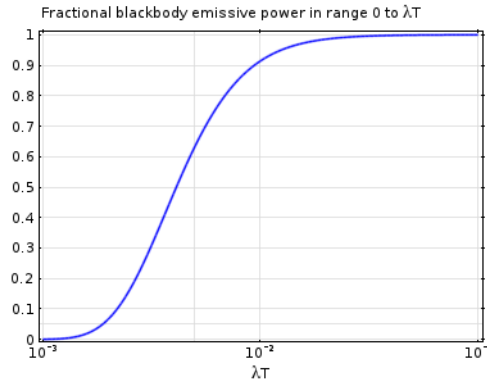


The function $e_b(T)$ is available as a predefined function via `ht.feb(T)` in the Heat Transfer interfaces.

Notice that:

$$F_{\lambda_1 T \rightarrow \lambda_2 T} = F_{0 \rightarrow \lambda_2 T} - F_{0 \rightarrow \lambda_1 T} \text{ and } F_{0 \rightarrow \infty} = 1$$

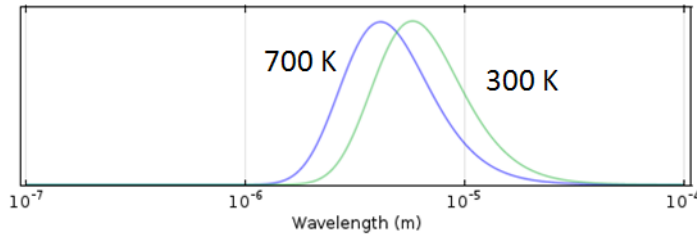
The figure below shows the value of $F_{0 \rightarrow \lambda T}$ for different values of λT .



DIFFUSE-GRAY SURFACES

Diffuse-gray surfaces correspond to the hypothesis that surface properties are independent of the radiation wavelength and angle between the surface normal and the radiation direction.

The assumption that the surface emissivity is independent of the radiation wavelength is often valid when most of the radiative power is concentrated on a relatively narrow spectral band. This is likely the case when the radiation is emitted by a surface at temperatures in limited range.



This setting is rarely applicable if there is solar radiation.

SOLAR AND AMBIENT SPECTRAL BANDS

When solar radiation is part of the model, it is possible to enhance a diffuse-gray surface model by considering two spectral bands: one for short wavelengths and one for large wavelengths.

It is interesting to notice that about 97% of the radiated power from a blackbody at 5800 K is at wavelengths of $2.5 \mu\text{m}$ or shorter, and 97% of the radiated power from a blackbody at 700 K is at wavelengths of $2.5 \mu\text{m}$ or longer (see [Figure 4-17](#)).

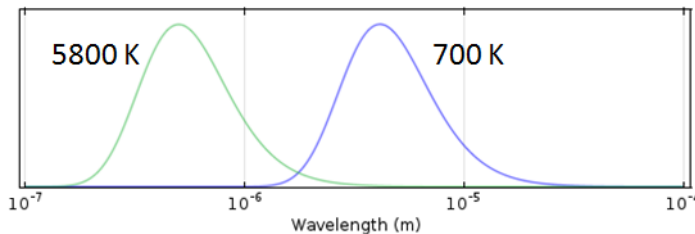


Figure 4-17: Normalized Planck distribution of blackbodies at 700 K and 5800 K.

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 $2.5 \mu\text{m}$
- An ambient band for wavelengths above $2.5 \mu\text{m}$

For each surface, properties are then described in terms of a solar absorptivity and an emissivity.

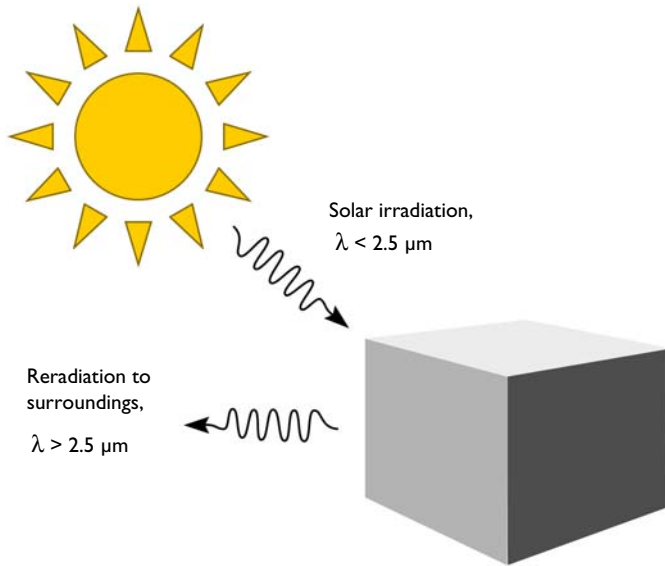


Figure 4-18: Absorption of solar radiation and emission to the surroundings.

By splitting the bands at the default of $2.5 \mu\text{m}$, 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.

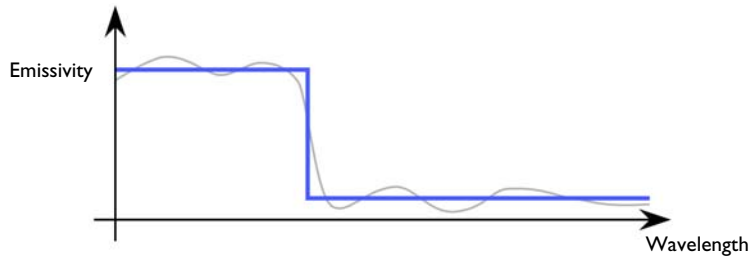
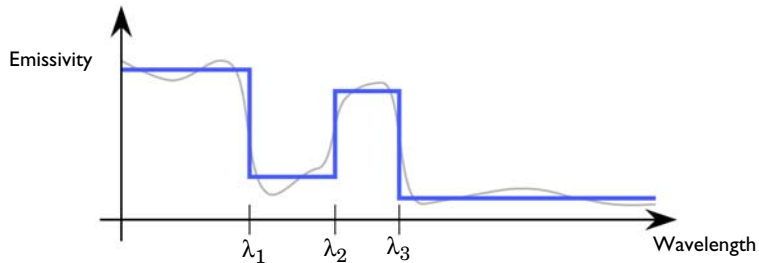


Figure 4-19: Solar and ambient spectral band approximation of the surface emissivity by a constant per band emissivity.

GENERAL DIFFUSE-SPECTRAL SURFACES

Diffuse-spectral surfaces correspond to the hypothesis that surface properties are wavelength dependent but independent of the angle between the surface normal and the radiation direction.

The heat transfer module enables to define constant surface properties per spectral bands and to adjust spectral intervals endpoints.



The multiple spectral bands approach is used in cases when the surface emissivity varies significantly over the bands of interest.

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, θ and φ , are defined below:

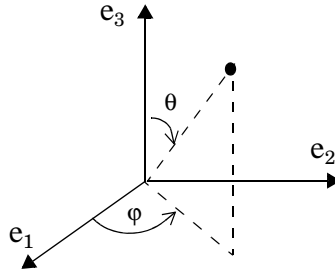


Figure 4-20: Definition of polar and azimuthal angles

The implementation supports directional surface properties that can be decomposed as the sum of two in two expressions, the first defined as a function of θ and φ only, and another one independent of θ and φ , but that may depend on any other model quantity like the temperature, or the spatial coordinates:

- Emissivity

$$\varepsilon_{\text{tot}}(\theta, \varphi, T, \mathbf{x}) = f_{\varepsilon}(\theta, \varphi) + \varepsilon(T, \mathbf{x})$$

- Transmissivity

$$\tau_{\text{tot}}(\theta, \varphi, T, \mathbf{x}) = f_{\tau}(\theta, \varphi) + \tau(T, \mathbf{x})$$

- Specular reflectivity

$$\rho_{\text{s, tot}}(\theta, \varphi, T, \mathbf{x}) = 1 - \rho_{\text{d}} - \varepsilon_{\text{tot}}(\theta, \varphi, T, \mathbf{x}) - \tau_{\text{tot}}(\theta, \varphi, T, \mathbf{x})$$

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 4-91, this source account for the difference between incident radiation, or *irradiation*, G , and radiation leaving the surface, or *radiosity*, J :

$$q = (1 - \rho_{\text{s}})G - J$$

The radiosity, J , is given in Equation 4-90. It is the sum of diffusely reflected and emitted radiation. For diffuse-gray surfaces, J is defined by:

$$J = \rho_d G + \epsilon e_b(T)$$

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 $0 \leq \epsilon \leq 1$. 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:

$$G = G_m + G_{\text{ext}} + G_{\text{amb}} \quad (4-105)$$

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:

$$G_{\text{ext}} = \sum F_{\text{ext}} P_s + \sum F_{\text{ext}} q_{0,s}$$

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:

$$G_{\text{amb}} = \epsilon_{\text{amb}} F_{\text{amb}} e_b(T_{\text{amb}})$$

- ϵ_{amb} is the *ambient emissivity*; (SI unit: 1), a dimensionless number in the range $0 \leq \epsilon_{\text{amb}} \leq 1$. $\epsilon_{\text{amb}} < 1$ 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, $0 \leq F_{\text{amb}} \leq 1$ 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:

$$J = \rho_{\text{d}}(G_{\text{m}}(J) + G_{\text{ext}} + G_{\text{amb}}) + \epsilon e_{\text{b}}(T) \quad (4-106)$$

- [Diffuse Mirror \(Surface-to-Surface Radiation Interface\)](#) is a variant of the Diffuse Surface radiation type with $\epsilon = 0$. 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. 20](#)). 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 $\epsilon e_{\text{b}}(T)$. 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

$$\epsilon + \rho_{\text{d}} + \rho_{\text{s}} = 1,$$

and the radiosity reads as in [Equation 4-106](#).

- [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

$$\epsilon + \rho_{\text{d}} + \rho_{\text{s}} + \tau = 1,$$

and the radiosities read

$$J_u = (\rho_{d,u}(G_{m,u}(J_u) + G_{ext,u} + G_{amb,u}) + \varepsilon_u e_b(T)) + \tau_d(G_{m,d}(J_d) + G_{ext,d} + G_{amb,d}) \quad (4-107)$$

$$J_d = (\rho_{d,d}(G_{m,d}(J_d) + G_{ext,d} + G_{amb,d}) + \varepsilon_d e_b(T)) + \tau_d(G_{m,u}(J_u) + G_{ext,u} + G_{amb,u}) \quad (4-108)$$

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 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: $W/(m^3 \cdot sr)$) defined in [Equation 4-104](#).

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),

$$\begin{cases} B_i = [\lambda_{i-1}, \lambda_i] & \text{for } 1 \leq i \leq N \\ \lambda_0 = 0 \\ \lambda_N = \infty \end{cases}$$

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} :

$$G_{\text{amb},i} = \int_{\lambda=\lambda_{i-1}}^{\lambda_i} G_{\text{amb}}(\lambda) d\lambda = F_{\lambda_{i-1}T \rightarrow \lambda_i T} F_{\text{amb},i} \varepsilon_{\text{amb},i} e_b(T_{\text{amb}})$$

- 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 :

$$G_{\text{ext},i} = \int_{\lambda=\lambda_{i-1}}^{\lambda_i} G_{\text{ext}}(\lambda) d\lambda = F_{\text{ext},i}(i_s) q_{0,s,i}$$

or

$$G_{\text{ext},i} = \int_{\lambda=\lambda_{i-1}}^{\lambda_i} G_{\text{ext}}(\lambda) d\lambda = F_{\text{ext},i}(i_s) P_{s,i}$$

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:

$$G_{\text{ext},i} = F_{\text{ext},i} F_{\lambda_{i-1}T \rightarrow \lambda_i T}(i_s) q_{0,s}$$

or

$$G_{\text{ext},i} = F_{\text{ext},i} F_{\lambda_{i-1}T \rightarrow \lambda_i T}(i_s) P_s$$

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:

$$J_i = \rho_{d,i}(G_{m,i}(J_i) + G_{\text{ext},i} + G_{\text{amb},i}) + \varepsilon_i e_b(T) \quad (4-109)$$

- **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. 20). 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 $F_{\lambda_1 T \rightarrow \lambda_2 T} e_b(T)$. 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 4-109.

- **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

$$J_{u,i} = \rho_{d,u,i}(G_{m,u,i}(J_{u,i}) + G_{\text{ext},u,i} + G_{\text{amb},u,i}) + \varepsilon_{u,i} e_b(T) + \tau_{d,i}(G_{m,d,i}(J_{d,i}) + G_{\text{ext},d,i} + G_{\text{amb},d,i}) \quad (4-110)$$

$$J_{d,i} = \rho_{d,u,i}(G_{m,d,i}(J_{d,i}) + G_{\text{ext},d,i} + G_{\text{amb},d,i}) + \varepsilon_{d,i} e_b(T) + \tau_{u,i}(G_{m,u,i}(J_{u,i}) + G_{\text{ext},u,i} + G_{\text{amb},u,i}) \quad (4-111)$$

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 4-106 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} .

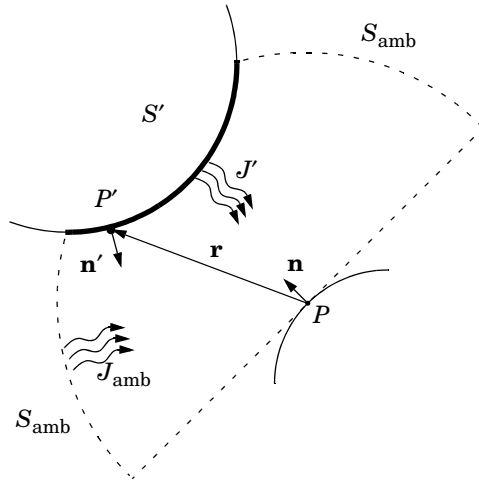


Figure 4-21: Example geometry for surface-to-surface radiation.

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' :

$$F_{\text{amb}} = 1 - F' = 1 - \int_{S'} \frac{(-\mathbf{n}' \cdot \mathbf{r})(\mathbf{n} \cdot \mathbf{r})}{\pi|\mathbf{r}|^4} ds$$

The two last equations plug into Equation 4-105 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

$$G_m = \int_{S_{\perp}'} \frac{(-\mathbf{n}' \cdot \mathbf{r}_{\perp})(\mathbf{n} \cdot \mathbf{r}_{\perp})}{2|\mathbf{r}_{\perp}|^3} J' ds \quad (4-112)$$

$$F_{\text{amb}} = 1 - \int_{S_{\perp}'} \frac{(-\mathbf{n}' \cdot \mathbf{r}_{\perp})(\mathbf{n} \cdot \mathbf{r}_{\perp})}{2|\mathbf{r}_{\perp}|^3} ds$$

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 4-112. For cases with specularly reflective surfaces, specular view factors depends also on specular reflectivities of surrounding surfaces, see Ref. 22.

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
- Discrete 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. 19](#). The estimated solar position is accurate for a date between year 2000 and 2199, due to an approximation used in the Julian Day calendar calculation.

The zenith angle, θ_s , and azimuth angle, φ_s , of the Sun are converted into a direction vector $\mathbf{i}_s = (i_{sx}, i_{sy}, i_{sz})$ in Cartesian coordinates assuming that the north, the west, and the up directions correspond to the x , y , and z directions, respectively, in the model. The relation between θ_s , φ_s , and \mathbf{i}_s is given by:

$$\begin{aligned} i_{sx} &= -\cos(\varphi_s) \sin(\theta_s) \\ i_{sy} &= \sin(\varphi_s) \sin(\theta_s) \\ i_{sz} &= -\cos(\theta_s) \end{aligned}$$

RADIATION IN AXISYMMETRIC GEOMETRIES

For an axisymmetric geometry, G_m and F_{amb} must be evaluated in a corresponding 3D geometry obtained by revolving the 2D boundaries around the axis. COMSOL Multiphysics creates this virtual 3D geometry by revolving the 2D boundary mesh into a 3D mesh. The resolution can be controlled in the azimuthal direction by setting the number of azimuthal sectors, which is the same as the number of elements to a full revolution. Try to balance this number against the mesh resolution in the rz -plane. This number, the azimuthal sectors, is accessible from the Radiation Settings section in physics interfaces for heat transfer.

Select between the hemicube and the direct area integration methods also in axial symmetry. Their settings work the same way as in 3D.



While G_m and F_{amb} are in fact evaluated in a full 3D, the number of points where they are requested is limited to the quadrature points on the boundary of a 2D geometry. The savings compared to a full 3D simulation are therefore substantial despite the full 3D view factor code being used.

Theory for Radiation in Participating Media

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

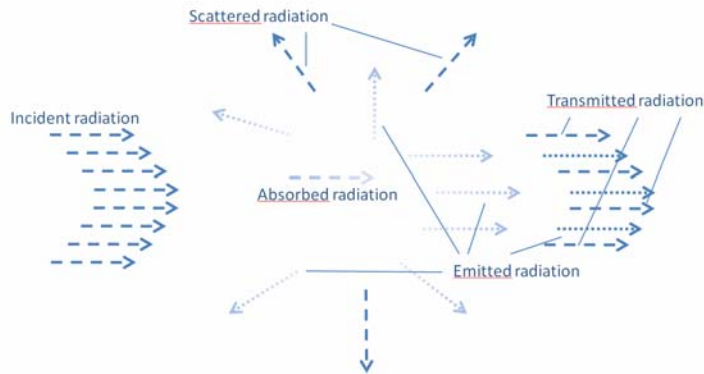


Figure 4-22: Example of interactions between participating media and radiation.

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. 22](#)):

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega' \quad (4-113)$$

where

- $I(\Omega)$ is the radiative intensity at a given position following the Ω direction (SI unit: $\text{W}/(\text{m}^2 \cdot \text{sr})$)
- $I_b(T)$ is the blackbody radiative intensity (SI unit: $\text{W}/(\text{m}^2 \cdot \text{sr})$), defined as

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi} \quad (4-114)$$



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: $\text{W}/(\text{m}^2 \cdot \text{K}^4)$)
- κ , β , σ_s are absorption, extinction, and scattering coefficients, respectively (SI unit: $1/\text{m}$) 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:

$$\phi(\mu_0) = 1 + \alpha_1 \mu_0$$

- *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 n th order Legendre polynomials.

Legendre polynomials can be defined by the Rodriguez formula:

$$P_k(x) = \frac{1}{2^k k!} \frac{d^k}{dx^k} ((x^2 - 1)^k)$$

- *Henye-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

$$I(\Omega) = \epsilon I_b(T) + \frac{1 - \epsilon}{\pi} q_{r, out} \quad \text{for all } \Omega \text{ such that } \mathbf{n} \cdot \Omega < 0$$

where

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi} \tag{4-115}$$

- Equation 4-115 is the blackbody radiation intensity and n_T is the refractive index
- ϵ is the surface emissivity, which is in the range $[0, 1]$
- $1 - \epsilon$ is the diffusive reflectivity
- \mathbf{n} is the outward normal vector
- $q_{r, \text{out}}$ is the heat flux coming from the domain and striking the wall:

$$q_{r, \text{out}} = \int_{\mathbf{n} \cdot \Omega > 0} I(\Omega)(\mathbf{n} \cdot \Omega) d\Omega$$

For black walls $\epsilon = 1$. 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

$$q_{r, \text{net}} = q_{r, \text{in}} - q_{r, \text{out}}$$

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

$$I(\Omega) = \epsilon I_b(T) + \frac{\rho_d}{\pi} q_{r, \text{out}} + \frac{\tau_d}{\pi} q_{r, \text{in, ext}} + \tau_s I_{\text{ext}}$$

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,
- $q_{r, \text{in}}$ is the heat flux striking the wall from exterior:

$$q_{r, \text{in, ext}} = \int_{\mathbf{n} \cdot \Omega \leq 0} I_{\text{ext}}(\Omega)(\mathbf{n} \cdot \Omega) d\Omega$$

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

$$q_{r, \text{net}} = q_{r, \text{in}} - q_{r, \text{out}} + \varepsilon_{\text{ext}} q_{r, \text{in, net}} - \varepsilon_{\text{ext}} \pi \varepsilon I_b(T)$$

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. 22](#)):

$$Q_r = \nabla \cdot \mathbf{q}_r = \kappa(G - 4\pi I_b(T))$$

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:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot \mathbf{q} = \kappa(G - 4n\sigma T^4) + \alpha_p T \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right) + \tau: \nabla \mathbf{u} + Q$$



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. 23) and yield S_N approximations with $N(N+2)$ directions in 3D or $N(N+2)/2$ in 2D (see Ref. 22). The quasi-uniform quadrature set yields T_N approximations with $8N^2$ directions in 3D or $4N^2$ in 2D (Ref. 44, Ref. 45). 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, \dots, 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 i th 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, \dots, I_n :

$$I_{i, \text{bnd}} = \varepsilon I_b(T) + \frac{1-\varepsilon}{\pi} q_{r, \text{out}} \quad \text{for all } \mathbf{S}_i \text{ such that } \mathbf{n} \cdot \mathbf{S}_i < 0 \quad (4-116)$$

with

$$q_{r, \text{out}} = \sum_{\mathbf{n} \cdot \Omega_j > 0} w_j I_j \mathbf{n} \cdot \Omega_j$$

In [Equation 4-116](#), the first term in the right hand side is the emitted radiative intensity, while the second term represents the reflected radiative intensity.

SEMITRANSSPARENT SURFACE

The node [Semitransparent Surface \(Radiation in Participating Media Interface\)](#) defines the following boundary condition for radiative intensities I_1, I_2, \dots, I_n :

$$I_{i, \text{bnd}} = \varepsilon I_b(T) + \frac{\rho_d}{\pi} q_{r, \text{out}} + \frac{\tau_d}{\pi} q_{r, \text{in, ext}} + \tau_s I_{i, \text{ext}} \quad (4-117)$$

for all \mathbf{S}_i such that $\mathbf{n} \cdot \mathbf{S}_i < 0$, with

$$q_{r, \text{out}} = \sum_{\mathbf{n} \cdot \Omega_j > 0} w_j I_j \mathbf{n} \cdot \Omega_j$$

and

$$q_{r, \text{in, ext}} = - \sum_{\mathbf{n} \cdot \Omega_j \leq 0} w_j I_{j, \text{ext}} \mathbf{n} \cdot \Omega_j$$

In [Equation 4-117](#), 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:

$$I_{i, \text{bnd}} = I_{\text{ext}} \quad \text{for all } \mathbf{S}_i \text{ such that } \mathbf{n} \cdot \mathbf{S}_i < 0$$

RADIATIVE SOURCE

The node **Radiative Source** accounts for a directional power density I_i in the radiative transfer equation:

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^n w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) + q_i$$

Discrete Ordinates Method Implementation in 2D

For a given index i , define two indices, i^+ and i^- , so that

- Ω , \mathbf{S}_{i^+} , and \mathbf{S}_{i^-} have the same components in the xy -plane
- and \mathbf{S}_{i^+} and \mathbf{S}_{i^-} have opposite components in the z direction.

Assuming that a model is invariant in the z direction, the radiative transfer equation in two directions, \mathbf{S}_{i^+} and \mathbf{S}_{i^-} , for the discrete ordinates method (DOM) reads:

$$\mathbf{S}_{i^+} \cdot \nabla I_{i^+} = \kappa I_b(T) - \beta I_{i^+} + \frac{\sigma_s}{4\pi} \sum_{j=1}^n w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i^+})$$

$$\mathbf{S}_{i^-} \cdot \nabla I_{i^-} = \kappa I_b(T) - \beta I_{i^-} + \frac{\sigma_s}{4\pi} \sum_{j=1}^n w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i^-})$$

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

$$\begin{aligned} \tilde{\mathbf{S}}_i \cdot \nabla \tilde{I}_i = \kappa I_b(T) - \beta \tilde{I}_i &+ \frac{\sigma_s}{8\pi} \sum_{j=1}^{n/2} w_j I_j (\phi(\mathbf{S}_j, \mathbf{S}_{i'}) + \phi(\mathbf{S}_j, \mathbf{S}_i)) \\ &+ \frac{\sigma_s}{8\pi} \sum_{j=1}^{n/2} w_j I_j (\phi(\mathbf{S}_j, \mathbf{S}_{i'}) + \phi(\mathbf{S}_j, \mathbf{S}_i)) \end{aligned}$$

or

$$\begin{aligned} \tilde{\mathbf{S}}_i \cdot \nabla \tilde{I}_i = \kappa I_b(T) - \beta \tilde{I}_i &+ \frac{\sigma_s}{8\pi} \sum_{j=1}^{n/2} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i'}) + w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \\ &+ \frac{\sigma_s}{8\pi} \sum_{j=1}^{n/2} w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_{i'}) + w_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \end{aligned}$$

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:

$$\tilde{\mathbf{S}}_i \cdot \nabla \tilde{I}_i = \kappa I_b(T) - \beta \tilde{I}_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^{n/2} \tilde{w}_j \tilde{I}_j \phi(\tilde{\mathbf{S}}_j, \tilde{\mathbf{S}}_i) \quad (4-118)$$

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

$$\int_{4\pi} I(\Omega)d\Omega \approx \sum_{j=1}^n w_j I_j = \sum_{j=1}^{n/2} w_j I_j + w_{j'} I_{j'} = \sum_{j=1}^{n/2} \tilde{w}_j \tilde{I}_j \quad (4-119)$$

with $\tilde{w}_j = 2w_j$.

Using results from Equation 4-118 and Equation 4-119 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.

PI Approximation Theory

The **PI approximation** is available as a radiation discretization method in [The Radiation in Participating Media Interface](#).

The PI 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 PI 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 PI 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. 22):

$$\nabla \cdot (-D_{P1} \nabla G) = -Q_r \quad (4-120)$$

where

- D_{P1} is the P1 diffusion coefficient, defined as

$$D_{P1} = \frac{1}{3\kappa + \sigma_s(3 - a_1)}$$

- a_1 is the linear Legendre coefficient of the scattering phase function
- Q_r is the radiative heat source:

$$Q_r = \kappa(G - 4\pi I_b) \quad (4-121)$$

When scattering is modeled as isotropic, $a_1=0$ and the P1 diffusion coefficient reduces to

$$D_{P1} = \frac{1}{3\kappa + 3\sigma_s}$$

The following boundary condition applies (Ref. 22):

$$\mathbf{n} \cdot (-D_{P1} \nabla G) = -q_{r, \text{net}}$$

where $q_{r, \text{net}}$ 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 4-120](#) is implemented.

In addition Q_r , defined by [Equation 4-121](#), is added as an heat source in the heat transfer equation:

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) + \nabla \cdot \mathbf{q} = Q_r + \alpha_p T \left(\frac{\partial p}{\partial t} + \mathbf{u} \cdot \nabla p \right) + \tau : \nabla \mathbf{u} + Q$$

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:

$$\mathbf{n} \cdot (-D_{P1} \nabla G) = q_{r, \text{net}}$$

The Opaque Surface feature accounts for the net radiative heat flux, $\mathbf{q}_{r, \text{net}}$, 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 :

$$q_{r, \text{net}} = \frac{\varepsilon}{2(2 - \varepsilon)} (4\pi I_{b, w} - G)$$

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

$$I_{b, w} = I_b = \frac{n^2 \sigma T^4}{\pi}$$

SEMITRANSSPARENT SURFACE

The [Semitransparent Surface \(Radiation in Participating Media Interface\)](#) 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, $\mathbf{q}_{r, \text{net}}$, accounted for in the heat balance, is defined as

$$q_{r, \text{net}} = \frac{2}{1 + \rho_d} (\pi \varepsilon I_{b, w} + \tau_d G) - \frac{1 - \rho_d}{2(1 + \rho_d)} G$$

where ρ_d is the surface diffuse reflectivity, and τ_d is the surface diffuse transmissivity.

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. On these boundaries, the relation between G (incident radiation), $\mathbf{q}_{r, \text{net}}$ (net radiative heat flux) and I_{ext} (incident radiative intensity) is

$$G + 2q_{r, \text{net}} = 4 \int_{\Omega \cdot \mathbf{n} < 0} I_{\text{ext}}(\Omega) \Omega \cdot \mathbf{n} d\Omega$$

by defining

$$I_{\text{ext}} = \int_{\Omega \cdot \mathbf{n} < 0} I_{\text{ext}}(\Omega) \Omega \cdot \mathbf{n} d\Omega$$

there is

$$q_{r, \text{net}} = \frac{1}{2}(4\pi I_{\text{ext}} - G)$$

which defines the heat radiative heat flux and also contributes to G boundary condition:

$$\mathbf{n} \cdot (-D_{\text{P1}} \nabla G) = -q_{r, \text{net}}$$

RADIATIVE SOURCE

The node [Radiative Source](#) accounts for a power density Q in the P1 approximation:

$$\nabla \cdot (-D_{\text{P1}} \nabla G) = -Q_{\text{r}} + Q.$$

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. 22](#)):

$$\Omega \cdot \nabla I(\Omega) = -\beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega' \quad (4-122)$$

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 4-122](#), 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 i th discrete ordinate.
- I_i is the i th component of the radiative intensity.
- ω_j is the j th quadrature weight.

If the [P1 Approximation Theory](#) is used instead, G is the solution of the following equation,

$$\nabla \cdot (-D_{P1} \nabla G) = -\kappa G, \quad (4-123)$$

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. 22) to account for wavelength dependence of emission, absorption and scattering through material properties, see the section [Wavelength Dependence of Surface Emissivity and Absorptivity](#).

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 Emissivity and Absorptivity](#).

If the [P1 Approximation Theory](#) is used instead, the incident radiation G_k for the spectral band k is solution of the following equation,

$$\nabla \cdot (-D_{\text{P1},k} \nabla G_k) = -Q_{r,k}, \quad (4-124)$$

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:

$$Q_{r,k} = \kappa_k (G_k - 4\pi I_b \text{FEP}_k(T)). \quad (4-125)$$

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 i^{th} 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 i^{th} 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. 22):

$$q_{r,\lambda} = -\frac{4\pi}{\beta_\lambda} \nabla i_{b,\lambda}$$

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:

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w = G \quad (4-126)$$

$$\mathbf{g}_w = -M_v D \nabla c_v$$

$$c_v = \phi_w c_{\text{sat}}$$

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/m³) is the vapor saturation concentration.
- D (SI unit: m²/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/(m²·s)) is the vapor diffusive flux.
- G (SI unit: kg/(m³·s)) is the moisture source (or sink).

CONCENTRATED SPECIES FORMULATION

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:

$$\rho_g \frac{\partial \omega_v}{\partial t} + \rho_g \mathbf{u} \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w = G \quad (4-127)$$

$$\mathbf{g}_w = -\rho_g D \nabla \omega_v$$

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 4-126, 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 the 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:

$$\frac{\partial w(\phi_w)}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w + \mathbf{u}_l \cdot \nabla \rho_l + \nabla \cdot \mathbf{g}_{lc} = G \quad (4-128)$$

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

$$\omega_v = \frac{M_v \phi_w c_{\text{sat}}(T)}{\rho_g}$$

- 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:

$$\mathbf{g}_w = -\rho_g D_{\text{eff}} \nabla \omega_v$$

- 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/m³) 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: m²/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/(m³·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:

$$G_{\text{evap}} = \frac{\partial[\varepsilon_p \rho_g \omega_v (1 - s_l)]}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w$$

The corresponding latent heat source is defined as:

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

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. 15 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

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot \left(\mathbf{g}_w - D_w \xi \nabla \phi_w \right) = G$$

Finally, the diffusive flux for vapor can be written as:

$$\mathbf{g}_w = -\rho_g D_{\text{eff}} \nabla \omega_v = -\rho_g D_{\text{eff}} \nabla \left[\frac{M_a M_v}{(x_a M_a + x_v M_v)^2} \frac{p_v}{p_A} \right]$$

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:

$$\mathbf{g}_w = -\frac{M_v D_{\text{eff}}}{R_{H_2O}} \frac{M_a}{(x_a M_a + x_v M_v)^2} \nabla p_v$$

The vapor permeability coefficient is then defined as

$$\delta_p = \frac{M_v D_{\text{eff}}}{R_{H_2O}} \frac{M_a}{(x_a M_a + x_v M_v)^2}$$

The Moisture Transport in Building Materials Interface solves for the following equation derived from Ref. 15:

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (-\xi D_w \nabla \phi_w - \delta_p \nabla (\phi_w p_{\text{sat}}(T))) = G \quad (4-129)$$

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:

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

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: $\text{kg}/\text{m}^3 \cdot \text{s}$) 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 Local Thermal Nonequilibrium 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.

THE NONISOTHERMAL FLOW AND CONJUGATE HEAT TRANSFER EQUATIONS

In industrial applications it is common that the density of a process fluid varies. These variations can have a number of different sources but the most common one is the presence of an inhomogeneous temperature field. This module includes the 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:

$$\begin{aligned}\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\ \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} &= -\nabla p + \nabla \cdot \boldsymbol{\tau} + \mathbf{F}\end{aligned}\tag{4-130}$$

where

- ρ is the density (SI unit: kg/m³)
- \mathbf{u} is the velocity vector (SI unit: m/s)
- p is the pressure (SI unit: Pa)
- $\boldsymbol{\tau}$ is the viscous stress tensor (SI unit: Pa), equal for a compressible fluid to:

$$\boldsymbol{\tau} = \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}$$

- μ is the dynamic viscosity (SI unit: Pa·s)
- \mathbf{F} is the body force vector (SI unit: N/m³)

It also solves the heat equation, which for a fluid is given in [Equation 4-18](#) by

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

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/m²)

- \mathbf{q}_r is the heat flux by radiation (SI unit: W/m^2)
- α_p is the coefficient of thermal expansion (SI unit: $1/\text{K}$):

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

- 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

$$Q_{vd} = \boldsymbol{\tau} : \nabla \mathbf{u}$$

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](#)):

$$\rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u}_{\text{trans}} \cdot \nabla T \right) + \nabla \cdot (\mathbf{q} + \mathbf{q}_r) = Q_{\text{ted}} + Q$$

where Q_{ted} is the thermoelastic damping heat source (SI unit: $\text{W}/(\text{m}^3)$). 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 \bar{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

$$\begin{aligned} \frac{\partial}{\partial t} \left(\bar{\rho} \left(\tilde{E} + \frac{\tilde{u}_i \tilde{u}_i}{2} \right) + \frac{\overline{\rho u_i'' u_i''}}{2} \right) + \frac{\partial}{\partial x_j} \left(\bar{\rho} \tilde{u}_j \left(\tilde{H} + \frac{\tilde{u}_i \tilde{u}_i}{2} \right) + \tilde{u}_j \frac{\overline{\rho u_i'' u_i''}}{2} \right) = \quad (4-131) \\ \frac{\partial}{\partial x_j} \left(-q_j - \overline{\rho u_j'' H''} + \tau_{ij} u_i'' - \frac{\overline{\rho u_j'' u_i'' u_i''}}{2} \right) + \frac{\partial}{\partial x_j} \left(\tilde{u}_i (\tau_{ij} - \overline{\rho u_i'' u_j''}) \right) \end{aligned}$$

where H is the enthalpy. The vector

$$q_j = -\lambda \frac{\partial T}{\partial x_j} \quad (4-132)$$

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*:

$$-\overline{\rho u_i'' u_j''} = \bar{\rho} \tau_{ij}^T = \mu_T \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu_T \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} - \frac{2}{3} \bar{\rho} k \delta_{ij} \quad (4-133)$$

where k is the turbulent kinetic energy, which in turn is defined by

$$\bar{\rho} k = \frac{1}{2} \overline{\rho u_i'' u_i''} \quad (4-134)$$

The correlation between u_j'' and H'' in Equation 4-131 is the turbulent transport of heat. It is modeled analogously to the laminar conductive heat flux

$$\overline{\rho u_j'' H''} = q_j^T = -\lambda_T \frac{\partial \tilde{T}}{\partial x_j} = -\frac{\mu_T C_p}{\text{Pr}_T} \frac{\partial \tilde{T}}{\partial x_j} \quad (4-135)$$

The molecular diffusion term,

$$\overline{\tau_{ij}u_i''}$$

and turbulent transport term,

$$\frac{\overline{\rho u_j'' u_i'' u_i''}}{2}$$

are modeled by a generalization of the molecular diffusion and turbulent transport terms found in the incompressible k equation

$$\overline{\tau_{ij}u_i''} - \frac{\overline{\rho u_j'' u_i'' u_i''}}{2} = \left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \quad (4-136)$$

Inserting Equation 4-132, Equation 4-133, Equation 4-134, Equation 4-135 and Equation 4-136 into Equation 4-131 gives

$$\begin{aligned} \frac{\partial}{\partial t} \left(\bar{\rho} \left(\tilde{E} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k \right) \right) + \frac{\partial}{\partial x_j} \left(\bar{\rho} \tilde{u}_j \left(\tilde{H} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k \right) \right) = \\ \frac{\partial}{\partial x_j} \left(-q_j - q_j^T + \left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left(\tilde{u}_i (\bar{\tau}_{ij} + \bar{\rho} \tau_{ij}^T) \right) \end{aligned} \quad (4-137)$$

The Favre average can also be applied to the momentum equation, which, using Equation 4-133, can be written

$$\frac{\partial}{\partial t} (\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j \tilde{u}_i) = - \frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} (\bar{\tau}_{ij} + \bar{\rho} \tau_{ij}^T) \quad (4-138)$$

Taking the inner product between \tilde{u}_i and Equation 4-138 results in an equation for the resolved kinetic energy, which can be subtracted from Equation 4-137 with the following result:

$$\begin{aligned} \frac{\partial}{\partial t} (\bar{\rho} (\tilde{E} + k)) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j (\tilde{E} + k)) = \\ - \bar{p} \frac{\partial \tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j} \left(-q_j - q_j^T + \left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left(\tilde{u}_i (\bar{\tau}_{ij} + \bar{\rho} \tau_{ij}^T) \right) \end{aligned} \quad (4-139)$$

where the relation

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

has been used.

According to Wilcox (Ref. 26), 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 4-139:

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E}) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial\tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}(-q_j - q_j^T) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\bar{\tau}_{ij} + \bar{\rho}\tau_{ij}^T)) \quad (4-140)$$

Larsson (Ref. 27) suggests to make the split

$$\bar{\tau}_{ij} = \tilde{\tau}_{ij} + \bar{\tau}_{ij}''$$

Since

$$\tilde{\tau}_{ij} \gg \bar{\tau}_{ij}''$$

for all applications of engineering interest, it follows that

$$\bar{\tau}_{ij} \approx \tilde{\tau}_{ij}$$

and consequently

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E}) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial\tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}\left((\lambda + \lambda_T)\frac{\partial\tilde{T}}{\partial x_j}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i\tilde{\tau}_{ij}^{\text{tot}}) \quad (4-141)$$

where

$$\tilde{\tau}_{ij}^{\text{tot}} = (\mu + \mu_T)\left(\frac{\partial\tilde{u}_i}{\partial x_j} + \frac{\partial\tilde{u}_j}{\partial x_i} - \frac{2}{3}\frac{\partial\tilde{u}_k}{\partial x_k}\delta_{ij}\right)$$

Equation 4-141 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. 27):

$$\bar{\rho}C_p\frac{\partial\tilde{T}}{\partial t} + \bar{\rho}C_p\tilde{u}_j\frac{\partial\tilde{T}}{\partial x_j} + \frac{\partial}{\partial x_j}\left(-(\lambda + \lambda_T)\frac{\partial\tilde{T}}{\partial x_j}\right) = -\frac{1}{\rho}\left(\frac{\partial\bar{p}}{\partial T}\right)_p\tilde{T}\left(\frac{\partial\bar{p}}{\partial t} + \tilde{u}_j\frac{\partial\bar{p}}{\partial x_j}\right) + \tilde{\tau}_{ij}\frac{\partial\tilde{u}_i}{\partial x_j}$$

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 Pr_T , while still quite simple. In [Ref. 28](#), it is compared to other models for 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

$$Pr_T = \left(\frac{1}{2Pr_{T\infty}} + \frac{0.3C_p\mu_T}{\lambda\sqrt{Pr_{T\infty}}} - \left(\frac{0.3C_p\mu_T}{\lambda} \right)^2 \left(1 - \exp\left(-\frac{\lambda}{0.3C_p\mu_T\sqrt{Pr_{T\infty}}} \right) \right) \right)^{-1} \quad (4-142)$$

where the Prandtl number at infinity is $Pr_{T\infty} = 0.85$ and λ is the conductivity.

Extended Kays-Crawford Weigand and others ([Ref. 29](#)) suggested an extension of [Equation 4-142](#) to liquid metals by introducing

$$Pr_{T\infty} = 0.85 + \frac{100\lambda}{C_p\mu Re_\infty^{0.888}}$$

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:

$$q_{wf} = \frac{\rho C_p u_\tau (T_w - T_f)}{T^+}$$

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. 30](#)):

$$T^+ = \begin{cases} \text{Pr}\delta_w^+ & \text{for } \delta_w^+ < \delta_{w1}^+ \\ 15\text{Pr}^{2/3} - \frac{500}{\delta_w^{+2}} & \text{for } \delta_{w1}^+ \leq \delta_w^+ < \delta_{w2}^+ \\ \frac{\text{Pr}_T}{\kappa} \ln \delta_w^+ + \beta & \text{for } \delta_{w2}^+ \leq \delta_w^+ \end{cases}$$

where in turn

$$\begin{aligned} \delta_w^+ &= \frac{\delta_w \rho \sqrt{C_\mu^{1/2} k}}{\mu} & \delta_{w1}^+ &= \frac{10}{\text{Pr}^{1/3}} \\ \delta_{w2}^+ &= 10 \sqrt{10 \frac{\kappa}{\text{Pr}_T}} & \text{Pr} &= \frac{C_p \mu}{\lambda} \\ \beta &= 15\text{Pr}^{2/3} - \frac{\text{Pr}_T}{2\kappa} \left(1 + \ln \left(1000 \frac{\kappa}{\text{Pr}_T} \right) \right) \end{aligned}$$

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:

$$[H_0]_-^+ = 0 \quad (4-143)$$

where H_0 is the total enthalpy.



- See [Screen](#) for the feature node details.
- Also see [Screen](#) boundary condition described for the single-phase flow interfaces.

THEORY FOR THE INTERIOR FAN BOUNDARY CONDITION

When the 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:

$$\left(\int_{\Gamma} \text{down}(\mathbf{u} \cdot \mathbf{n} \rho C_p) dS \right) T_{\text{avg}} = \int_{\Gamma} \text{down}(\mathbf{u} \cdot \mathbf{n} \rho C_p T) dS$$

- If the direction is opposite to normal vector, the outlet temperature T_{avg} is defined by:

$$\left(\int_{\Gamma} \text{up}(\mathbf{u} \cdot \mathbf{n} \rho C_p) dS \right) T_{\text{avg}} = \int_{\Gamma} \text{up}(\mathbf{u} \cdot \mathbf{n} \rho C_p T) dS$$

Theory for the Moisture Flow Interface

The following points of the theory of Moisture Flow are discussed in this part:

- [The Moisture Flow Equations](#)
- [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

The density and viscosity of moist air depend on its moisture content. 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.

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:

$$\begin{aligned}\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\ \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} &= -\nabla p + \nabla \cdot \boldsymbol{\tau} + \mathbf{F}\end{aligned}\tag{4-144}$$

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)
- $\boldsymbol{\tau}$ is the viscous stress tensor (SI unit: Pa), equal for a compressible fluid to:

$$\boldsymbol{\tau} = \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}$$

- 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 equation for moisture transport in air, given in [Equation 4-126](#) by

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g} = G$$

where in addition to the quantities above

- 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} is the moisture flux by diffusion (SI unit: $\text{kg}/(\text{m}^2 \cdot \text{s})$):

$$\mathbf{g} = -M_v D \nabla c_v$$

- D (SI unit: m^2/s) is the vapor diffusion coefficient in air.
- G contains moisture sources (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$)

TURBULENT MOISTURE FLOW THEORY

The Favre average \tilde{c}_v of the vapor concentration c_v is defined by

$$\tilde{c}_v = \frac{\overline{\rho c_v}}{\bar{\rho}}$$

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:

$$M_v \frac{\partial \tilde{c}_v}{\partial t} + M_v \tilde{u}_j \frac{\partial \tilde{c}_v}{\partial x_j} + M_v \frac{\partial}{\partial x_j} \left(-(D + D_T) \frac{\partial \tilde{c}_v}{\partial x_j} \right) = G \quad (4-145)$$

Kays-Crawford Model for Turbulent Diffusivity

By analogy with turbulent heat transfer, the turbulent Schmidt number Sc_T is given by (Ref. 28)

$$Sc_T = \left(\frac{1}{2Sc_{T\infty}} + \frac{0.3\mu_T Sc}{\mu \sqrt{Sc_{T\infty}}} - \left(0.3 \frac{\mu_T Sc}{\mu} \right)^2 \left(1 - \exp\left(-\frac{\mu}{0.3\mu_T Sc \sqrt{Sc_{T\infty}}} \right) \right) \right)^{-1} \quad (4-146)$$

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. 30) 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 concentration $c_{v,a}$ and a wall with vapor concentration $c_{v,w}$, is:

$$g_{wf} = \frac{M_V u_\tau (c_{v,w} - c_{v,a})}{\phi_w^+} \cong \frac{M_V u_\tau c_{sat} (\phi_{w,w} - \phi_{w,a})}{\phi_w^+}$$

where u_τ is the friction velocity, c_{sat} is the saturation concentration at the lift-off position, and ϕ_w^+ is the dimensionless relative humidity given by (Ref. 30):

$$\phi_w^+ = \begin{cases} Sc \delta_w^+ & \text{for } \delta_w^+ < \delta_{w1}^+ \\ 15 Sc^{2/3} - \frac{500}{\delta_w^{+2}} & \text{for } \delta_{w1}^+ \leq \delta_w^+ < \delta_{w2}^+ \\ \frac{Sc_T}{\kappa} \ln \delta_w^+ + \beta & \text{for } \delta_{w2}^+ \leq \delta_w^+ \end{cases}$$

where in turn

$$\begin{aligned} \delta_w^+ &= \frac{\delta_w \rho \sqrt{C_\mu^{1/2} k}}{\mu} & \delta_{w1}^+ &= \frac{10}{Sc^{1/3}} \\ \delta_{w2}^+ &= 10 \sqrt{10 \frac{\kappa}{Sc_T}} & Sc &= \frac{\mu}{\rho D} \\ \beta &= 15 Sc^{2/3} - \frac{Sc_T}{2\kappa} \left(1 + \ln \left(1000 \frac{\kappa}{Sc_T} \right) \right) \end{aligned}$$

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:

$$[c_v]_{-}^{+} = 0 \quad (4-147)$$

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:

$$\left(\int_{\Gamma} \text{down}(\rho \mathbf{u} \cdot \mathbf{n}) dS \right) c_{avg} = \int_{\Gamma} \text{down}(\rho \mathbf{u} \cdot \mathbf{n} c_v) dS$$

- If the direction is opposite to normal vector, the outlet vapor concentration c_{avg} is defined by:

$$\left(\int_{\Gamma} \text{up}(\rho \mathbf{u} \cdot \mathbf{n}) dS \right) c_{avg} = \int_{\Gamma} \text{up}(\rho \mathbf{u} \cdot \mathbf{n} c_v) dS$$

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:

$$P = ST$$

$$\mu_{\text{Th}} = T \frac{dS}{dT}$$

where P is the Peltier coefficient (SI unit: V), S is the Seebeck coefficient (SI unit: V/K), T is the temperature (SI unit: K), and μ_{Th} is the Thomson coefficient (SI unit: V/K). These relations show that all coefficients can be considered different descriptions of one and the same quantity. The COMSOL formulation primarily uses the Seebeck coefficient. The Peltier coefficient is also used as an intermediate variable, but the Thomson coefficient is not used.

When simulating the thermoelectric effect, the following fluxes are the quantities of interest:

- Conductive heat flux \mathbf{q} , defined by

$$\mathbf{q} = -k\nabla T + P\mathbf{J} \quad (4-148)$$

- Electric current density \mathbf{J} , defined by

$$\mathbf{J} = -\sigma(\nabla V + S\nabla T) \quad (4-149)$$

Thermoelectric efficiency is measured by the figure of merit Z (SI unit: 1/K), defined as:

$$Z = \frac{S^2\sigma}{k}$$

where σ is the electrical conductivity and k the thermal conductivity.

Some other quantities of relevance are the electric field \mathbf{E} and the Joule heat source Q :

$$\mathbf{E} = -\nabla V$$

$$Q = \mathbf{J} \cdot \mathbf{E}$$

From these definitions, conservation of heat energy and electrical current in an immobile solid reads:

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

$$\nabla \cdot \mathbf{J} = Q_j$$

where ρ is the density, C_p the heat capacity, and Q_j is the current source.

HOW THE SEEBECK, PELTIER, AND THOMSON EFFECTS ARE INCLUDED IN THE GENERAL FORMULATION

The general formulation of thermoelectric effect redefines the heat flux and the electric current according to [Equation 4-148](#) and [Equation 4-149](#), 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 4-149](#) used in the general formulation.

Peltier Effect

The Peltier effect is described as the conversion of 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 \mathbf{q} term in the heat equation when \mathbf{q} is defined following [Equation 4-148](#).

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 \mathbf{q} is defined following Equation 4-148. This time consider the term $-\mathbf{T}\mathbf{J} \cdot \nabla S$. Assuming that S is function of T , then:

$$-\mathbf{T}\mathbf{J} \cdot \nabla S = -T \frac{dS}{dT} \mathbf{J} \cdot \nabla T = -\mu_{\text{Th}} \mathbf{J} \cdot \nabla T$$

Theory for the Local Thermal Nonequilibrium Interface

The detailed theory leading to the equations of local thermal nonequilibrium heat transfer in porous media is presented above in [Theory for Heat Transfer in Porous Media](#). This part only recalls the main results and describes how [Local Thermal Nonequilibrium](#) multiphysics coupling feature implements them.

The local thermal nonequilibrium hypothesis describes heat transfer in a porous medium using two temperature fields to solve: T_f for the fluid phase and T_s for the porous matrix. These should satisfy the following couple of partial differential equations:

$$\theta_s \rho_s C_{p,s} \frac{\partial T_s}{\partial t} + \nabla \cdot \mathbf{q}_s = q_{sf}(T_f - T_s) + \theta_s \mathbf{Q}_s \quad (4-150)$$

$$\varepsilon_p \rho_f C_{p,f} \frac{\partial T_f}{\partial t} + \rho_f C_{p,f} \mathbf{u}_p \cdot \nabla T_f + \nabla \cdot \mathbf{q}_f = q_{sf}(T_s - T_f) + \varepsilon_p \mathbf{Q}_f \quad (4-151)$$

Recall the Fourier's law of conduction adapted to the local thermal nonequilibrium hypothesis:

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

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

and the quantities used in this problem:

- θ_s is the solid volume fraction (SI unit: 1)
- ε_p is the porosity (SI unit: 1)
- ρ_s and ρ_f are the solid and fluid densities (SI unit: kg/m^3)
- $C_{p,s}$ and $C_{p,f}$ are the solid and fluid heat capacities at constant pressure (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$)
- \mathbf{q}_s and \mathbf{q}_f are the solid and fluid conductive heat fluxes (SI unit: W/m^2)

- k_s and k_f are the solid and fluid thermal conductivities (SI unit: W/(m·K))
- q_{sf} is the interstitial convective heat transfer coefficient (SI unit: W/(m³·K))
- Q_s and Q_f are the solid and fluid heat sources (SI unit: W/m³)
- \mathbf{u}_p is the porous velocity vector (SI unit: m/s)

PREDEFINED MULTIPHYSICS INTERFACE

The [Local Thermal Nonequilibrium Interface](#) is a predefined coupling between [The Heat Transfer in Solids Interface](#) and [The Heat Transfer in Fluids Interface](#). These two interfaces solve for [Equation 4-150](#) and [Equation 4-151](#), respectively, but without the heat exchange term $\pm q_{sf}(T_f - T_s)$.

The [Local Thermal Nonequilibrium](#) multiphysics coupling feature combines two actions in order to couple the two aforementioned physics interfaces. It first multiplies each energy equation by its volume fraction: θ_s and ϵ_p for solid and fluid phases, respectively. Then it adds the heat exchange term $\pm q_{sf}(T_f - T_s)$ in both equations.

VOLUMETRIC AND SURFACE THERMAL CONDITIONS

As shown in [Equation 4-150](#) and [Equation 4-151](#), the volumetric heat sources $\theta_s Q_s$ and $\epsilon_p Q_f$ are applied to the energy equations. The [Heat Source](#) features of each physics interface though specifies Q_s and Q_f . Special care is therefore needed when defining a heat source for the whole porous medium. You would have to ensure that the heat source densities, Q_s and Q_f , are both equal to the heat rate density that was intended to the porous medium.

Theory for the 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. 15](#):

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot (-k_{\text{eff}} \nabla T - L_v \delta_p \nabla(\phi_w p_{\text{sat}})) = Q \quad (4-152)$$

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (-\xi D_w \nabla \phi_w - \delta_p \nabla(\phi_w p_{\text{sat}})) = G \quad (4-153)$$

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:

$$\rho_g C_p \frac{\partial T}{\partial t} + \rho_g C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (4-154)$$

$$\mathbf{q} = -k \nabla T \quad (4-155)$$

$$Q = -(C_{p,v} - C_{p,a}) \mathbf{g}_w \quad (4-156)$$

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w = G \quad (4-157)$$

$$\mathbf{g}_w = -M_v D \nabla c_v \quad (4-158)$$

$$c_v = \phi_w c_{\text{sat}} \quad (4-159)$$

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:

$$q_{\text{evap}} = L_v g_{\text{evap}} \quad (4-160)$$

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:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + (\rho_g C_{p,g} \mathbf{u}_g + \rho_l C_{p,l} \mathbf{u}_l) \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + Q_{\text{evap}} \quad (4-161)$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \quad (4-162)$$

$$Q = -[(C_{p,v} - C_{p,a}) \mathbf{g}_w + C_{p,l} \mathbf{g}_{lc}] \quad (4-163)$$

$$Q_{\text{evap}} = L_v G_{\text{evap}} \quad (4-164)$$

$$\frac{\partial w(\phi_w)}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w + \mathbf{u}_l \cdot \nabla \rho_l + \nabla \cdot \mathbf{g}_{lc} = G \quad (4-165)$$

$$\mathbf{g}_w = -\rho_g D_{\text{eff}} \nabla \omega_v \quad (4-166)$$

$$G_{\text{evap}} = \frac{\partial[\varepsilon_p \rho_g \omega_v (1 - s_l)]}{\partial t} + \rho_g \mathbf{u}_g \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w \quad (4-167)$$

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](#).

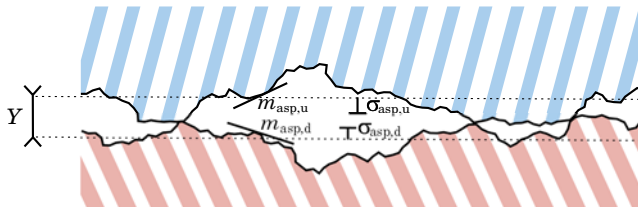


Figure 4-23: Contacting surfaces at the microscopic level.

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](#)):

$$\sigma_{asp} = \sqrt{\sigma_{u, asp}^2 + \sigma_{d, asp}^2} \quad m_{asp} = \sqrt{m_{u, asp}^2 + m_{d, asp}^2}$$

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:

$$h_c = 1.25k_{\text{contact}} \frac{m_{\text{asp}}}{\sigma_{\text{asp}}} \left(\frac{p}{H_c} \right)^{0.95}$$

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:

$$k_{\text{contact}} = \frac{2k_u k_d}{k_u + k_d}$$



When k_u (resp. k_d) is not isotropic, it is replaced by its normal conductivity $\mathbf{n}^T k_u \mathbf{n}$ (resp. $\mathbf{n}^T k_d \mathbf{n}$).

The relative pressure p/H_c can be evaluated by specifying H_c directly or using the following relation (4.16.1 in Ref. 1) for the relative pressure using c_1 and c_2 , the Vickers correlation coefficient and size index:

$$\frac{p}{H_c} = \left(\frac{p}{c_1 \left(1.62 \frac{\sigma_{\text{asp}}}{\sigma_0} m_{\text{asp}} \right)^{c_2}} \right)^{\frac{1}{(1+0.071c_2)}}$$

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):

$$\frac{c_1}{H_0} = 4.0 - 5.77 \frac{H_B}{H_0} + 4.0 \left(\frac{H_B}{H_0} \right)^2 - 0.61 \left(\frac{H_B}{H_0} \right)^3$$

$$c_2 = -0.37 + 0.442 \frac{H_B}{c_1}$$

The Brinell hardness is denoted by H_B , and H_0 is equal to 3.178 GPa.

Mikic Elastic Correlation

The Mikic correlation is valid for isotropic rough surfaces and assumes elastic deformations of surface asperities. It gives h_c by the following relation:

$$h_c = 1.54 k_{\text{contact}} \frac{m_{\text{asp}}}{\sigma_{\text{asp}}} \left(\frac{\sqrt{2} p}{m E_{\text{contact}}} \right)^{0.94}$$

Here, E_{contact} is an effective Young's modulus for the contact interface, satisfying (4.16.3 in Ref. 1):

$$\frac{1}{E_{\text{contact}}} = \frac{1 - \nu_u^2}{E_u} + \frac{1 - \nu_d^2}{E_d}$$

where E_u and E_d are the Young's moduli of the two contacting surfaces and ν_u and ν_d are the Poisson's ratios.

GAP CONDUCTANCE

The gap conductance due to interstitial fluid cannot be neglected for high fluid thermal conductivity or high contact pressure. The parallel-plate gap gas correlation assumes that the interstitial fluid is a gas and defines h_g by:

$$h_g = \frac{k_g}{Y + M_g}$$

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:

$$M_g = \alpha \beta \Lambda \quad \Lambda = \frac{k_B T_g}{\sqrt{2} \pi D^2 p_g}$$

In these relations, α is the contact thermal accommodation parameter, β is a gas property parameter (equal to 1.7 for air), Λ is the gas mean free path, k_B is the Boltzmann constant, D is the average gas particle diameter, p_g is the gas pressure (often the atmospheric pressure), and T_g is the gap temperature equal to:

$$T_g = \frac{T_u + T_d}{2}$$

The mean separation thickness, Y , is a function of the contact pressure, p . For low values of p near 0 Pa, Y goes to infinity since no contact occur. For high values of p — greater than $H_c/2$ in the Cooper-Mikic-Yovanovich model and greater than $H_c/4$ in the Mikic elastic model — Y reduces to 0 meaning that the contact is considered as perfect.

RADIATIVE CONDUCTANCE

At high temperatures, above 600°C, radiative conductance needs to be considered. The gray-diffuse parallel plate model provides the following formula for h_r :

$$h_r = \frac{\varepsilon_u \varepsilon_d}{\varepsilon_u + \varepsilon_d - \varepsilon_u \varepsilon_d} \sigma (T_u^3 + T_u^2 T_d + T_u T_d^2 + T_d^3)$$

which implies that:

$$h_r(T_u - T_d) = \frac{\varepsilon_u \varepsilon_d}{\varepsilon_u + \varepsilon_d - \varepsilon_u \varepsilon_d} \sigma (T_u^4 - T_d^4)$$

$$h_r(T_d - T_u) = \frac{\varepsilon_u \varepsilon_d}{\varepsilon_u + \varepsilon_d - \varepsilon_u \varepsilon_d} \sigma (T_d^4 - T_u^4)$$

THERMAL FRICTION

The friction heat, 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:

$$r = \frac{1}{1 + \xi_d} \quad \xi_d = \sqrt{\frac{\rho_u C_{p,u} k_u}{\rho_d C_{p,d} k_d}}$$

and symmetrically, $(1-r)$ is:

$$(1-r) = \frac{1}{1 + \xi_u} \quad \xi_u = \sqrt{\frac{\rho_d C_{p,d} k_d}{\rho_u C_{p,u} k_u}}$$

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.

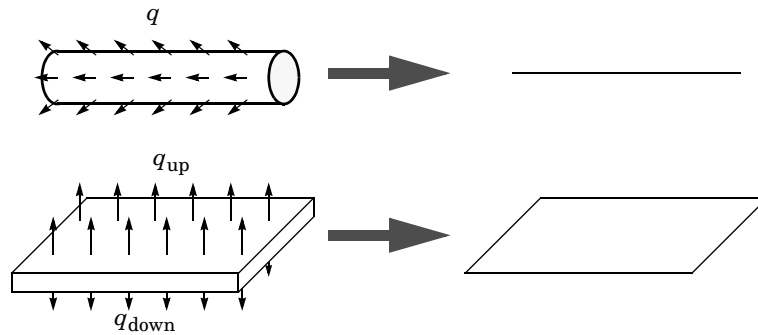


Figure 4-24: Geometry reduction from 3D to 1D (top) and from 3D to 2D (bottom).

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:

$$d_z \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = d_z Q + q_0 \quad (4-168)$$

$$d_z \rho C_p \frac{\partial T}{\partial t} + \rho C_p d_z \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = d_z Q + q_0 \quad (4-169)$$

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 6-17](#) is replaced by

$$(2\pi r d_z) \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = (2\pi r d_z) Q + q_0 \quad (4-170)$$

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

$$(2\pi r d_z) \rho C_p \frac{\partial T}{\partial t} + (2\pi r d_z) \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = (2\pi r d_z) Q + q_0 \quad (4-171)$$

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 6-17](#) is replaced by

$$A_c \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = A_c Q + q_0 \quad (4-172)$$

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

$$A_c \rho C_p \frac{\partial T}{\partial t} + A_c \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = A_c Q + q_0 \quad (4-173)$$

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.

The Heat Transfer Coefficients

One of the most common boundary conditions when modeling heat transfer is convective cooling or heating whereby a fluid cools or heats a surface by natural or forced convection. In principle, it is possible to model this process in two ways:

- Using a heat transfer coefficient on the surfaces
- Extending the model to describe the flow and heat transfer in the surrounding fluid

The second approach is the most accurate if the geometry or the external flow is complicated. The Heat Transfer Module includes the Conjugate Heat Transfer predefined multiphysics coupling and the CFD Module includes the 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.

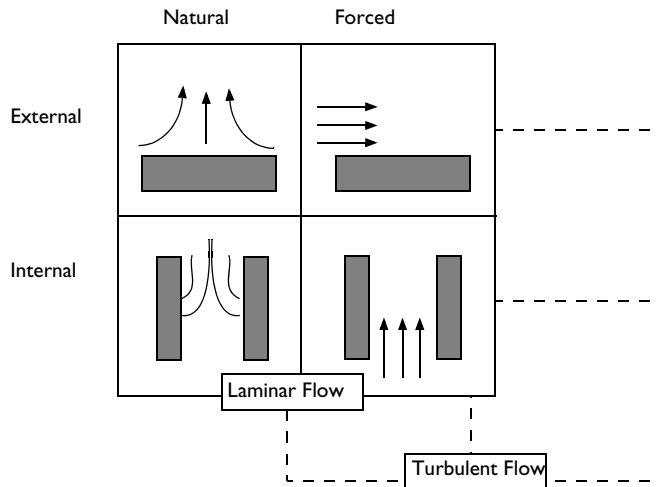


Figure 4-25: The eight categories of convective heat flux.

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, $Nu_L = hL/k$
- The Reynolds number, $Re_L = \rho UL/\mu$

- The Prandtl number, $Pr = \mu C_p / k$
- The Rayleigh number, $Ra_L = Gr_L Pr$

where

- h is the heat transfer coefficient (SI unit: $W/(m^2 \cdot K)$)
- 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: $W/(m \cdot K)$)
- 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: $Pa \cdot s$)
- C_p is the heat capacity at constant pressure of the fluid (SI unit: $J/(kg \cdot K)$)

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:

$$Gr_L = \frac{g\rho(\rho_{ext} - \rho_s)L^3}{\mu^2}$$

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 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):

$$\text{Gr}_L = \frac{g\alpha_p(T_s - T_{\text{ext}})L^3}{(\mu/\rho)^2}$$

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: $\text{Ra} = \text{Pr Gr}$. A small value of the Ra number indicates that conduction dominates. In such a 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

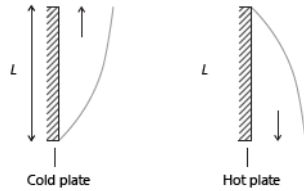


Figure 4-26: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a vertical wall.

The following correlations correspond to equations 9.26 and 9.27 in Ref. 20:

$$h = \begin{cases} \frac{k}{L} \left(0.68 + \frac{0.67 \text{Ra}_L^{1/4}}{\left(1 + \left(\frac{0.492k}{\mu C_p} \right)^{9/16} \right)^{4/9}} \right) & \text{if } \text{Ra}_L \leq 10^9 \\ \frac{k}{L} \left(0.825 + \frac{0.387 \text{Ra}_L^{1/6}}{\left(1 + \left(\frac{0.492k}{\mu C_p} \right)^{9/16} \right)^{8/27}} \right)^2 & \text{if } \text{Ra}_L > 10^9 \end{cases} \quad (4-174)$$

where the height of the wall, L , is a correlation input.

The Rayleigh number is defined as:

$$\text{Ra}_L = \frac{g \alpha_p \rho^2 C_p |T - T_{\text{ext}}| L^3}{k \mu} \quad (4-175)$$

if the density depends on temperature only, or as:

$$\text{Ra}_L = \frac{g \rho C_p |\rho_{\text{ext}} - \rho_s| L^3}{k \mu} \quad (4-176)$$

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 $10^4 \leq \text{Ra}_L \leq 10^{13}$.

INCLINED WALL

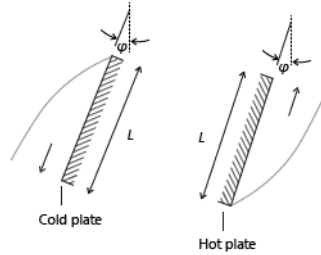


Figure 4-27: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on an inclined wall.

The following correlations correspond to equations 9.26 and 9.27 in Ref. 20 (the same as for a vertical wall):

$$h = \begin{cases} \frac{k}{L} \left(0.68 + \frac{0.67(\cos \varphi \text{Ra}_L)^{1/4}}{\left(1 + \left(\frac{0.492k}{\mu C_p}\right)^{9/16}\right)^{4/9}} \right) & \text{if } \text{Ra}_L \leq 10^9 \\ \frac{k}{L} \left(0.825 + \frac{0.387\text{Ra}_L^{1/6}}{\left(1 + \left(\frac{0.492k}{\mu C_p}\right)^{9/16}\right)^{8/27}} \right)^2 & \text{if } \text{Ra}_L > 10^9 \end{cases} \quad (4-177)$$

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 $-60^\circ < \varphi < 60^\circ$ and $10^4 \leq \text{Ra}_L \leq 10^{13}$.

The definition of the Raleigh number, Ra_L , is analogous to the one for vertical walls and is given by the following:

$$\text{Ra}_L = \frac{g \alpha_p \rho^2 C_p |T - T_{\text{ext}}| L^3}{k \mu} \quad (4-178)$$

if the density depends on temperature only, or as:

$$\text{Ra}_L = \frac{g\rho C_p |\rho_{\text{ext}} - \rho_s| L^3}{k\mu} \quad (4-179)$$

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 $\cos \phi$ in the expression for h , because this gives better accuracy (see Ref. 40).



According to Ref. 20, 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. 40). Unfortunately, little data is available about transition. There is some data available in Ref. 40 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 $(T + T_{\text{ext}})/2$, except ρ_s which is evaluated at the wall temperature, T .

HORIZONTAL PLATE, UPSIDE

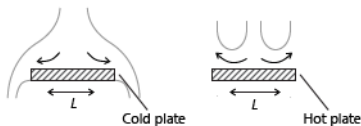


Figure 4-28: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on the top surface of a horizontal plate.

The following correlations correspond to equations 9.30–9.32 in Ref. 20 but can also be found as equations 7.77 and 7.78 in Ref. 40.

If $\rho < \rho_{\text{ext}}$, then

$$h = \begin{cases} \frac{k}{L} 0.54 \text{Ra}_L^{1/4} & \text{if } 10^4 \leq \text{Ra}_L \leq 10^7 \\ \frac{k}{L} 0.15 \text{Ra}_L^{1/3} & \text{if } 10^7 \leq \text{Ra}_L \leq 10^{11} \end{cases} \quad (4-180)$$

while if $\rho \geq \rho_{\text{ext}}$, then

$$h = \frac{k}{L} 0.27 \text{Ra}_L^{1/4} \text{ if } 10^5 \leq \text{Ra}_L \leq 10^{10} \quad (4-181)$$

Ra_L is given by Equation 4-175 or Equation 4-176, and L , the characteristic length (defined as area/perimeter, see Ref. 40) is a correlation input. The material data are evaluated at $(T + T_{\text{ext}})/2$, except ρ_s which is evaluated at the wall temperature, T .

When the density depends only on temperature, the conditions $\rho < \rho_{\text{ext}}$ and $\rho \geq \rho_{\text{ext}}$ can be replaced by $T > T_{\text{ext}}$ and $T \leq T_{\text{ext}}$ respectively.

HORIZONTAL PLATE, DOWNSIDE

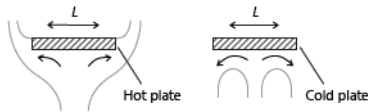


Figure 4-29: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on the bottom surface of an horizontal plate.

Equation 4-180 is used when $\rho \geq \rho_{\text{ext}}$ (or $T \leq T_{\text{ext}}$) and Equation 4-181 is used when $\rho < \rho_{\text{ext}}$ (or $T > T_{\text{ext}}$). Otherwise it is the same implementation as for Horizontal Plate, Upside.

LONG HORIZONTAL CYLINDER

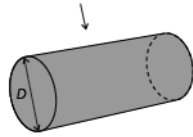


Figure 4-30: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a long horizontal cylinder.

The following correlations correspond to equations 9.34 in Ref. 20. It is validated for $Ra_D \leq 10^{12}$.

$$h = \frac{k}{D} \left(0.6 + \frac{0.387 Ra_D^{1/6}}{\left(1 + \left(\frac{0.559}{Pr} \right)^{9/16} \right)^{8/27}} \right)^2 \quad (4-182)$$

Here D is the cylinder diameter and Ra_D is given by

$$Ra_D = \frac{g \alpha_p \rho^2 C_p |T - T_{\text{ext}}| D^3}{k \mu}$$

if the density depends on temperature only, or as:

$$Ra_D = \frac{g \rho C_p |\rho_{\text{ext}} - \rho_s| D^3}{k \mu}$$

if the density depends on other parameters.

The material data are evaluated at $(T + T_{\text{ext}})/2$, except ρ_s which is evaluated at the wall temperature, T .

SPHERE

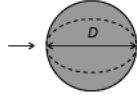


Figure 4-31: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a sphere.

The following correlations correspond to equation 9.35 in Ref. 20. It is validated for $Ra_D \leq 10^{11}$ and $Pr \geq 0.7$.

$$h = \frac{k}{D} \left(2 + \frac{0.589 Ra_D^{1/4}}{\left(1 + \left(\frac{0.469}{Pr} \right)^{9/16} \right)^{4/9}} \right) \quad (4-183)$$

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 .

VERTICAL THIN CYLINDER

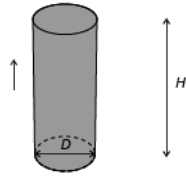


Figure 4-32: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a vertical thin cylinder.

The following correlation corresponds to equation 7.83 in Ref. 40. It is validated only for side walls of the thin cylinder ($\delta_T \geq D$), the horizontal disks (top and bottom) should be treated as horizontal plates. If the boundary thin layer is much smaller than D , vertical wall correlations should be used.

$$h = \frac{k}{H} \left(\frac{4}{3} \left(\frac{7 \text{Ra}_H \text{Pr}}{5(20 + 21 \text{Pr})} \right)^{1/4} + \frac{4(272 + 315 \text{Pr})H}{35(64 + 63 \text{Pr})D} \right)$$

where D is the cylinder diameter, H is the cylinder height, and Ra_H is given by

$$\text{Ra}_H = \frac{g \alpha_p |T - T_{\text{ext}}| H^3}{k_\mu}$$

if the density depends on temperature only, or as:

$$\text{Ra}_H = \frac{g \rho C_p |\rho_{\text{ext}} - \rho_s| H^3}{k_\mu}$$

if the density depends on other parameters.

The material data are evaluated at $(T + T_{\text{ext}})/2$, except ρ_s which is evaluated at the wall temperature, T .

NARROW CHIMNEY, PARALLEL PLATES

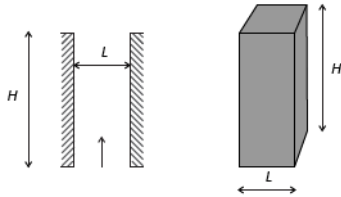


Figure 4-33: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection in a parallel-plate narrow chimney.

If $\text{Ra}_L < H/L$, then

$$h = \frac{k}{H} \frac{1}{24} \text{Ra}_L \quad (4-184)$$

where the plate distance, L , and the chimney height, H , are correlation inputs (equation 7.96 in Ref. 40). Ra_L is given by Equation 4-175 or Equation 4-176. The material data are evaluated at $(T + T_{\text{ext}})/2$.

NARROW CHIMNEY, CIRCULAR TUBE

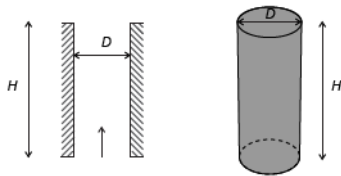


Figure 4-34: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection in a circular narrow chimney.

If $\text{Ra}_D < H/D$, then

$$h = \frac{k}{H} \frac{1}{128} \text{Ra}_D$$

where the tube diameter, D , and the chimney height, H , are correlation inputs (table 7.2 in Ref. 40 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 $(T + T_{\text{ext}})/2$.

Heat Transfer Coefficients — External Forced Convection

PLATE, AVERAGED TRANSFER COEFFICIENT

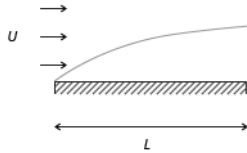


Figure 4-35: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a horizontal plate.

This correlation is a combination of equations 7.33 and 7.41 in Ref. 20:

$$h = \begin{cases} \frac{2k}{L} \frac{0.3387 \text{Pr}^{1/3} \text{Re}_L^{1/2}}{\left(1 + \left(\frac{0.0468}{\text{Pr}}\right)^{2/3}\right)^{1/4}} & \text{if } \text{Re}_L \leq 5 \cdot 10^5 \\ \frac{2k}{L} \text{Pr}^{1/3} (0.037 \text{Re}_L^{4/5} - 871) & \text{if } \text{Re}_L > 5 \cdot 10^5 \end{cases} \quad (4-185)$$

where $\text{Pr} = \mu C_p / k$ and $\text{Re}_L = \rho UL / \mu$. The plate length, L , and the exterior velocity, U , are correlation inputs. The material data are evaluated at $(T + T_{\text{ext}})/2$.

PLATE, LOCAL TRANSFER COEFFICIENT

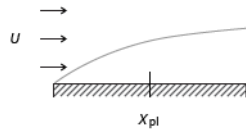


Figure 4-36: Schematic representation of geometry and parameters for the local heat transfer coefficient correlation applied to forced convection on a horizontal plate.

This correlation corresponds to equations 5.79b and 5.131' in Ref. 40:

$$h = \begin{cases} \frac{k}{x} 0.332 \text{Pr}^{1/3} \text{Re}_x^{1/2} & \text{if } \text{Re}_x \leq 5 \cdot 10^5 \\ \frac{k}{x} 0.0296 \text{Pr}^{1/3} \text{Re}_x^{4/5} & \text{if } \text{Re}_x > 5 \cdot 10^5 \end{cases} \quad (4-186)$$

where $\text{Pr} = \mu C_p / k$ and $\text{Re}_x = \rho U x / \mu$. 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 $k / \max(x, \sqrt{\epsilon})$ where ϵ is the floating-point relative accuracy.

CYLINDER IN CROSS-FLOW

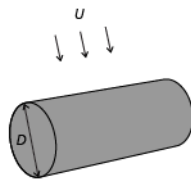


Figure 4-37: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a cylinder in cross-flow.

This correlation is given by equation 7.54 in Ref. 20:

$$h = \frac{k}{D} \left(0.3 + \frac{0.62 \text{Re}_D^{1/2} \text{Pr}^{1/3}}{\left(1 + \left(\frac{0.4}{\text{Pr}}\right)^{2/3}\right)^{1/4}} \left(1 + \left(\frac{\text{Re}_D}{282000}\right)^{5/8}\right)^{4/5} \right) \text{if } \text{Re}_D \text{Pr} \geq 0.2 \quad (4-187)$$

where $\text{Pr} = \mu C_p / k$ and $\text{Re}_D = \rho U D / \mu$. The cylinder diameter, D , and the exterior velocity, U , are correlation inputs. The material data are evaluated at $(T + T_{\text{ext}}) / 2$.

SPHERE

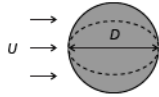


Figure 4-38: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a sphere.

This correlation corresponds to equation 7.56 in Ref. 20:

$$h = \frac{k}{D} \left(2 + (0.4 \text{Re}_D^{1/2} + 0.06 \text{Re}_D^{2/3}) \text{Pr}^{0.4} \left(\frac{\mu}{\mu_s} \right)^{1/4} \right) \quad (4-188)$$

$$\text{if} \left\{ \begin{array}{l} 3.5 \leq \text{Re}_D \leq 7.6 \cdot 10^4 \\ 0.71 \leq \text{Pr} \leq 380 \\ 1 \leq \frac{\mu}{\mu(T)} \leq 3.2 \end{array} \right.$$

where $\text{Pr} = \mu C_p / k$ and $\text{Re}_D = \rho U D / \mu$. 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 .

ISOTHERMAL TUBE

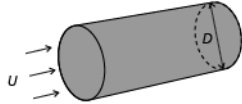


Figure 4-39: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection in an isothermal circular tube.

This correlation corresponds to equations 8.55 and 8.61 in Ref. 20:

$$h = \begin{cases} \frac{k}{D} 3.66 & \text{if } \text{Re}_D \leq 2500 \\ \frac{k}{D} 0.027 \text{Re}_D^{4/5} \text{Pr}^{1/3} \left(\frac{\mu}{\mu_s} \right)^{0.14} & \text{if } \text{Re}_D > 2500 \end{cases} \quad (4-189)$$

where $\text{Pr} = \mu C_p / k$, and $\text{Re}_D = \rho U_{\text{ext}} D / \mu$. 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 .

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

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\text{ext}} - T)$$

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

$$-\mathbf{n} \cdot \mathbf{g} = M_v h_m (c_{v, \text{ext}} - c_v) \quad (4-190)$$

where M_v is the molar mass of water vapor, h_m is a moisture transfer coefficient and $c_{v, \text{ext}}$ the vapor concentration of the external fluid far from the boundary.

In [Ref. 20](#), 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 :

$$h_m = \frac{DLe^n}{k} h = \frac{D}{k} \left(\frac{k}{\rho C_p D} \right)^n h \quad (4-191)$$

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. 20](#).

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](#).

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

$$Ra = \frac{\rho^2 g |\alpha_p| C_p \Delta T L^3}{\mu k}$$

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

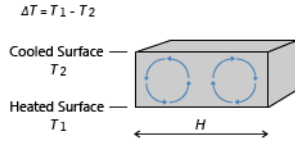


Figure 4-40: Schematic representation of geometry and parameters for the equivalent thermal conductivity correlation applied to an horizontal cavity heated from below.

The following correlation corresponds to equation 9.49 in Ref. 20. It is validated for $3 \cdot 10^5 \leq Ra_H \leq 7 \cdot 10^9$:

$$Nu = 0.069 Ra_H^{1/3} Pr^{0.074} \quad (4-192)$$

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 $Pr = \mu C_p / k$. The material data are evaluated at $(T_1 + T_2)/2$.

Vertical Cavity With Sidewall Heating

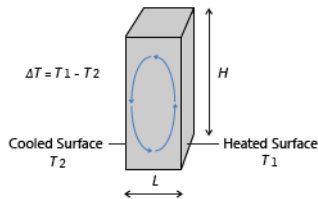


Figure 4-41: Schematic representation of geometry and parameters for the equivalent thermal conductivity correlation applied to a vertical cavity with sidewall heating.

The following correlations correspond to equations 9.50, 9.51, and 9.52 in Ref. 20:

- If $1 \leq H/L \leq 2$, $10^{-3} \leq Pr \leq 10^5$, and $10^3 \leq \frac{Pr Ra_L}{0.2 + Pr}$:

$$\text{Nu} = 0.18 \left(\frac{\text{Pr}}{0.2 + \text{Pr}} \text{Ra}_L \right)^{0.29} \quad (4-193)$$

- If $2 \leq H/L \leq 10$, $\text{Pr} \leq 10^5$, and $10^3 \leq \text{Ra}_L \leq 10^{10}$:

$$\text{Nu} = 0.22 \left(\frac{\text{Pr}}{0.2 + \text{Pr}} \text{Ra}_L \right)^{0.28} \left(\frac{H}{L} \right)^{-1/4} \quad (4-194)$$

- If $10 \leq H/L \leq 40$, $1 \leq \text{Pr} \leq 2 \cdot 10^4$, and $10^4 \leq \text{Ra}_L \leq 10^7$:

$$\text{Nu} = 0.42 \text{Ra}_L^{1/4} \text{Pr}^{0.012} \left(\frac{H}{L} \right)^{-0.3} \quad (4-195)$$

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 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. 41
Acetic Acid - Air	$-0.994 \cdot 10^{-4}$	$2.958 \cdot 10^{-2}$	Ref. 41
Ethanol - Air	$-0.832 \cdot 10^{-4}$	$2.405 \cdot 10^{-2}$	Ref. 41
Toluene - Air	$-1.189 \cdot 10^{-4}$	$3.09 \cdot 10^{-2}$	Ref. 41
Diethyl Ether - Air	$-0.908 \cdot 10^{-4}$	$1.892 \cdot 10^{-2}$	Ref. 41
Glycerol - Air	$-0.885 \cdot 10^{-4}$	$6.517 \cdot 10^{-2}$	Ref. 41
Heptane - Nitrogen	$-0.980 \cdot 10^{-4}$	$2.21 \cdot 10^{-2}$	Ref. 42
Mercury - Mercury (Vapor)	$-2.049 \cdot 10^{-4}$	$49.06 \cdot 10^{-2}$	Ref. 42
Ethylene Glycol - Ethylene Glycol (Vapor)	$-0.890 \cdot 10^{-4}$	$5.021 \cdot 10^{-2}$	Ref. 42

More data can be found in [Ref. 41](#) and [Ref. 42](#).

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](#)):

$$\mathbf{q}_{\text{tot}} = \rho \mathbf{u}E + \mathbf{q} + \mathbf{q}_r \quad (4-196)$$

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](#).

TOTAL ENERGY FLUX

The total energy flux is equal to:

$$\mathbf{e}_{\text{tot}} = \rho \mathbf{u} \left(\frac{1}{2} \mathbf{u} \cdot \mathbf{u} \right) + \rho \mathbf{u} (E + \Psi) + \mathbf{q} + \mathbf{q}_r - \sigma \mathbf{u}$$

Again, *convective* heat flux, $\rho \mathbf{u} E$, *conductive* heat flux, \mathbf{q} , and *radiative* heat flux, \mathbf{q}_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):

$$\mathbf{e}_{\text{tot}} = \rho \mathbf{u} \left(\frac{1}{2} \mathbf{u} \cdot \mathbf{u} \right) + \rho \mathbf{u} (E + \Psi) + \mathbf{q} + \mathbf{q}_r - (-p \mathbf{I} + \tau) \mathbf{u}$$

Introducing the total internal energy, E_0 , and total enthalpy, H_0 :

$$E_0 = E + \frac{1}{2} \mathbf{u} \cdot \mathbf{u} \quad H_0 = H + \frac{1}{2} \mathbf{u} \cdot \mathbf{u}$$

leads to the following equivalent expression involving total enthalpy:

$$\mathbf{e}_{\text{tot}} = \rho \mathbf{u} (H_0 + \Psi) + \mathbf{q} + \mathbf{q}_r - \tau \mathbf{u}$$

The potential Ψ has a simple form in some special cases — for example, for gravitational effects (Chapter 1.4 in Ref. 39) — but it is in general rather difficult to derive. Potential energy is therefore often excluded and the total energy flux is approximated by

$$\mathbf{e}_{\text{tot}} = \rho \mathbf{u} E_0 + \mathbf{q} + \mathbf{q}_r - \sigma \mathbf{u} \quad (4-197)$$

The total energy flux vector, \mathbf{e}_{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.

$$\frac{d}{dt} \int_{\Omega} \rho E dv + \int_{\partial\Omega_{\text{ext}}} \mathbf{q}_{\text{tot}} \cdot \mathbf{n} ds = Q_{\text{Int}} - W_{\text{str, Int}} \quad (4-198)$$

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 4-198 is more visually represented by the diagram of [Figure 4-42](#) below.

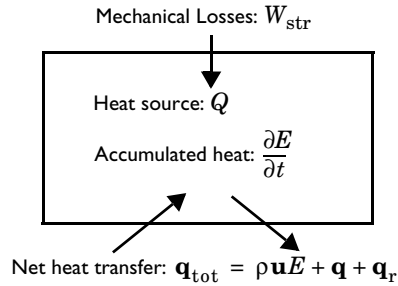


Figure 4-42: A heat balance diagram.

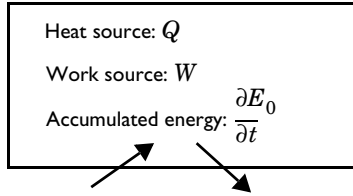
Several predefined variables are available in COMSOL Multiphysics to describe the heat rates involved in the system heat balance. See [Global Variables for Heat and Energy Balance](#) for their definition.

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](#)):

$$\frac{d}{dt} \int_{\Omega} \rho E_0 dv + \int_{\partial\Omega_{\text{ext}}} \mathbf{e}_{\text{tot}} \cdot \mathbf{n} ds = Q_{\text{Int}} + W_{\text{Int}} \quad (4-199)$$

The variables in this formula are defined in [Total Heat Flux and Energy Flux](#). Equation 4-199 is more visually represented by the diagram of [Figure 4-43](#) below.



Net energy transfer: $\mathbf{e}_{\text{tot}} = \rho \mathbf{u} E_0 + \mathbf{q} + \mathbf{q}_r - \sigma \mathbf{u}$

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*:

$$\mathbf{F} = \begin{bmatrix} \frac{\partial x}{\partial X} & \frac{\partial x}{\partial Y} & \frac{\partial x}{\partial Z} \\ \frac{\partial y}{\partial X} & \frac{\partial y}{\partial Y} & \frac{\partial y}{\partial Z} \\ \frac{\partial z}{\partial X} & \frac{\partial z}{\partial Y} & \frac{\partial z}{\partial Z} \end{bmatrix} \quad (4-200)$$

It relates elementary distances $d\mathbf{x}$ and $d\mathbf{X}$ in the domain, expressed in material and spatial frames, according to:

$$d\mathbf{x} = \mathbf{F}d\mathbf{X} \quad (4-201)$$

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: $\text{W}/(\text{m}^3 \cdot \text{K})$),
- Heat flux, q_0 (SI unit: W/m^2),
- Heat transfer coefficient, h (SI unit: $\text{W}/(\text{m}^2 \cdot \text{K})$).

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 4-201](#).

THERMAL CONDUCTIVITY

Thermal conductivity, k , is a tensor density. The relationship between the value on the spatial frame and the material frame is:

$$k_{(x, y, z)} = \frac{1}{\det(\mathbf{F})}\mathbf{F}k_{(X, Y, Z)}\mathbf{F}^T$$

With this relation, and recalling that

$$\mathbf{n}_{(X, Y, Z)} = \mathbf{F}^T\mathbf{n}_{(x, y, z)}$$

$$\nabla_{(X, Y, Z)}T = \mathbf{F}^T\nabla_{(x, y, z)}T$$

the total conductive heat flux through a boundary, computed in both frames according to the integrals below, gives the same result:

$$\int_{\partial\Omega_0} -k_{(X, Y, Z)}\nabla_{(X, Y, Z)}T \cdot \mathbf{n}_{(X, Y, Z)}ds_0 = \int_{\partial\Omega} -k_{(x, y, z)}\nabla_{(x, y, z)}T \cdot \mathbf{n}_{(x, y, z)}ds$$

Here, $\partial\Omega_0$ and $\partial\Omega$ are the boundaries of the same domain in material and spatial frames, respectively.

THERMAL CONDUCTIVITY OF A LAYER

The same transformations are applied to thermal conductivity but with different transformation matrices. The deformation gradient tensor depends on the layer type:

- When the layer is resistive, the deformation gradient tensor \mathbf{F}_{xdim} is equal to the deformation gradient tensor \mathbf{F} defined in [Equation 4-200](#).
- When the layer is conductive, the deformation gradient tensor $\mathbf{F}_{\mathbf{t}}$ 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:

$$\mathbf{F}_{\text{xdim}} = \begin{bmatrix} xT_X + n_x n_X & xT_Y + n_x n_Y & xT_Z + n_x n_Z \\ yT_X + n_y n_X & yT_Y + n_y n_Y & yT_Z + n_y n_Z \\ zT_X + n_z n_X & zT_Y + n_z n_Y & zT_Z + n_z n_Z \end{bmatrix}$$

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. Poinsoot 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. Hasegall, F. Di Gennaro, C. Baumgartner, E. Neufeld, M.C. Gosselin, D. Payne, A. Klingenböck, N. Kuster, *IT²IS 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. 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.
15. EN 15026, *Hygrothermal performance of building components and building elements - Assessment of moisture transfer by numerical simulation*, CEN, 2007.
16. J. Bear and Y. Bachmat, *Introduction to Modeling of Transport Phenomena in Porous Media*, Kluwer Academic Publisher, 1990.
17. H. Lee, *Thermal Design, Heat Sinks, Thermoelectrics, Heat Pipes, Compact Heat Exchangers, and Solar Cells*, John Wiley & Sons, 2006.
18. 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.
19. R. Siegel and J. Howell, *Thermal Radiation Heat Transfer*, 4th ed., Taylor & Francis, New York, 2002.
20. 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.
21. <https://www.esrl.noaa.gov/gmd/grad/solcalc/>
22. M.F. Modest, *Radiative Heat Transfer*, 2nd ed., Academic Press, San Diego, California, 2003.
23. W.A. Fiveland, "The Selection of Discrete Ordinate Quadrature Sets for Anisotropic Scattering," *Fundamentals of Radiation Transfer*, HTD, vol. 160, ASME, 1991.
24. A. Bejan et al., *Heat Transfer Handbook*, John Wiley & Sons, 2003.
25. F. Charron, *Partage de la chaleur entre deux corps frottants*, Publication Scientifique et Technique du Ministère de l'Air, no. 182, 1943. (In French)
26. D.C. Wilcox, *Turbulence Modeling for CFD*, 2nd ed., DCW Industries, 1998.

27. 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.
28. W.M. Kays, “Turbulent Prandtl Number — Where Are We?,” *ASME J. Heat Transfer*, vol. 116, pp. 284–295, 1994.
29. 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.
30. 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.
31. 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.
32. 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.
33. 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.
34. 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.
35. J.L. Monteith and M.H. Unsworth, *Principles of Environmental Physics*, Edward Arnold, London, 290 pp., 1990.
36. 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.
37. 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.
38. W. Wagner and H-J Kretschmar, *International Steam Tables*, 2nd ed., Springer, 2008.
39. M. Kaviany, *Principles of Convective Heat Transfer*, 2nd ed., Springer, 2001.

40. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1993.
41. 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
42. J.J. Jasper, “The Surface Tension of Pure Liquid Compounds”, *J. Phys. Chem. Ref. Data*, vol. 1, pp. 841–1010, 1972.
43. G.K. Batchelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.
44. 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.
45. M. A. Badri, P. Jolivet, B. Rousseau, S. Le Corre, H. Dignonnet 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.

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 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
Lumped Thermal System	lts	Lumped Thermal System

TABLE 5-1: THE HEAT TRANSFER MODULE INTERFACES

INTERFACE	TAG	VERSIONS
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](#) (http) 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 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 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 `ht1sh` 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.

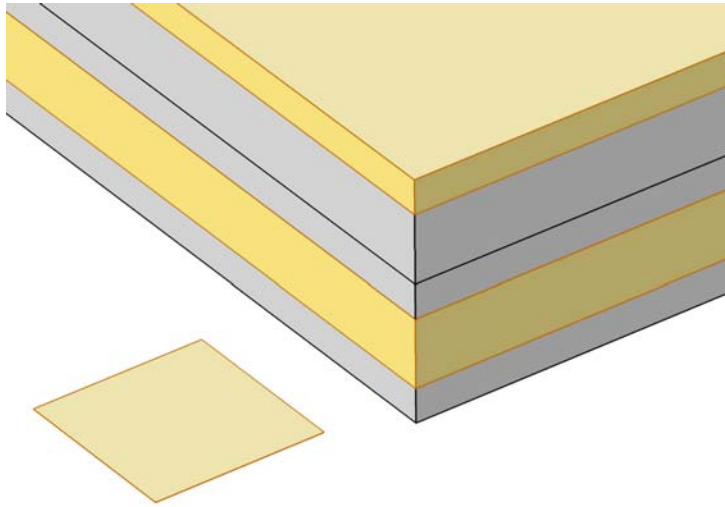


Figure 5-1: Boundary selection with layers 2 and 5 selected.

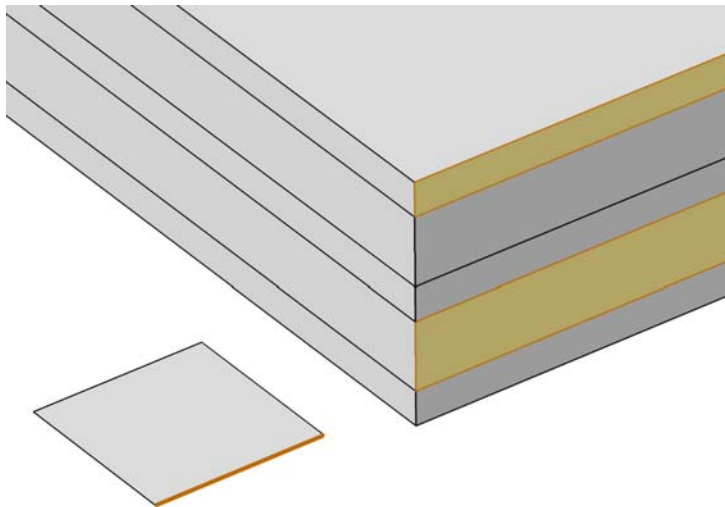


Figure 5-2: Edge selection with layers 2 and 5 selected.

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.

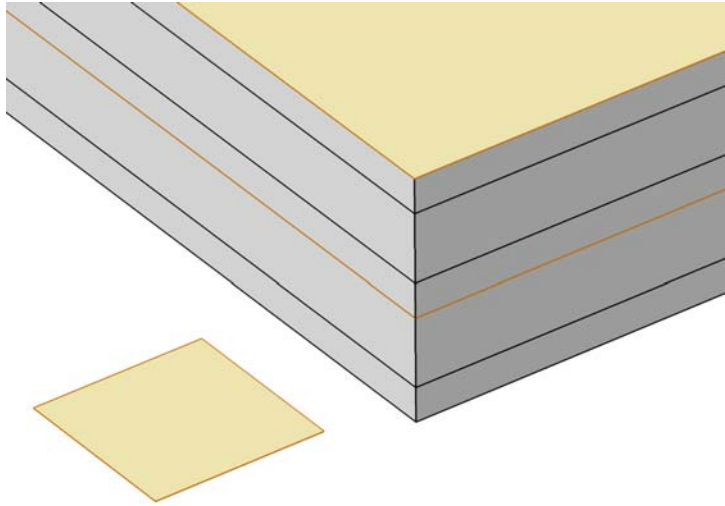


Figure 5-3: Boundary selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

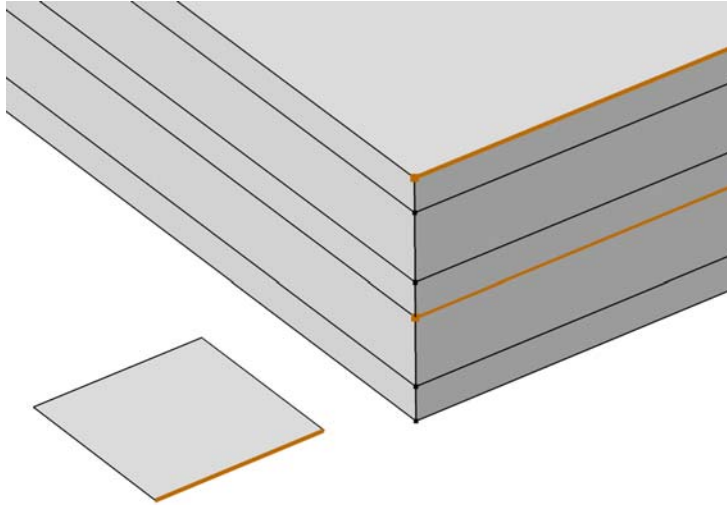


Figure 5-4: Edge selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

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.

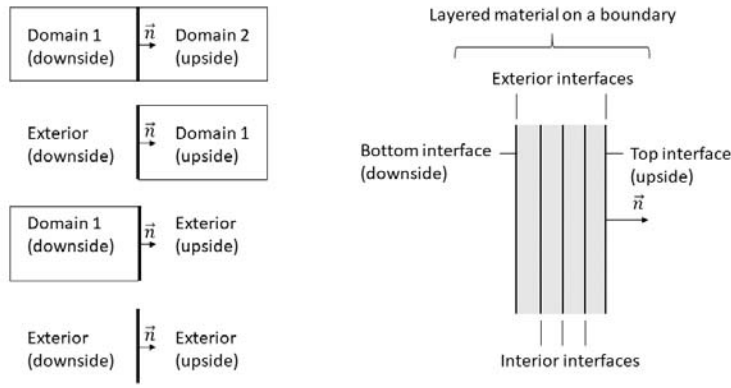


Figure 5-5: Definition of upside and downside settings for a boundary for different configurations, and corresponding interfaces for a layered material.



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_11mat1_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 γ Plus, 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	—
	Local Thermal Nonequilibrium	—
	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.



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.


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 $ht.T_{\text{ref}}$.

Select the **Isothermal domain** check box to activate the support for isothermal domain modeling. See [Isothermal Domain Interface](#) and [Isothermal Domain](#).



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

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/>

DOMAIN NODES FOR THE HEAT TRANSFER INTERFACE

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
- 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.

BOUNDARY NODES FOR THE HEAT TRANSFER INTERFACE

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
- 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.

EDGE NODES FOR THE HEAT TRANSFER INTERFACE

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

POINT NODES FOR THE HEAT TRANSFER INTERFACE

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 `ht1sh`.

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, s1mat1), 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.



- The layer thickness variable, `ht1sh.ds`, used in the weak equations, is the product of the thickness set in the **Shell Properties** section, `ht1sh.lth`, and the scale factor `ht1sh.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, `ht1sh.ds`, is defined in all dimensions, and is unrelated to the **Out-of-plane thickness** of the layered material, `ht1sh.d`, which is only available in 2D, and might be edited in the [Out-of-Plane Thickness](#) section.



- 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 4-70](#)). 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 `ht1sh.Tref`.



This input is not overridden 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.

CONSISTENT STABILIZATION

The **Streamline diffusion** check box is selected by default and should remain selected for optimal performance for heat transfer in fluids or other applications that include a convective or translational term. **Crosswind diffusion** provides extra diffusion in regions with sharp gradients. The added diffusion is orthogonal to the streamlines, so streamline diffusion and crosswind diffusion can be used simultaneously. The **Crosswind diffusion** check box is also selected by default.


INCONSISTENT STABILIZATION

The **Isotropic diffusion** check box is not selected by default.



Heat Transfer Consistent and Inconsistent Stabilization Methods

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](#)
- [Override and Contribution](#)

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.



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](#)
- [Moist Air \(Moisture Transport Interface\)](#)
- [Moisture Source](#)
- [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 () 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 () , found under the **Heat Transfer** branch () when adding a physics interface, is used to model heat transfer by conduction, convection, and radiation in porous media. A **Porous Medium** model is active by default on all domains. All functionality for including other domain types, such as a solid domain, is also available.

The temperature equation defined in porous media domains corresponds to the convection-diffusion equation with thermodynamic properties averaging models to account for both solid matrix and fluid properties. This equation is valid when the temperatures into the porous matrix and the fluid are in equilibrium. If not, use [The Local Thermal 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**.



See [Settings for the Heat Transfer Interface](#) and [Feature Nodes for the Heat Transfer Interface](#) for details.




[Theory for Heat Transfer in Porous Media](#)



Free Convection in a Porous Medium: Application Library path
Subsurface_Flow_Module/Heat_Transfer/convection_porous_medium

The Heat Transfer in Moist Porous Media Interface

The **Heat Transfer in Moist Porous Media (ht)** interface () 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 4-70](#)) 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.

DEPENDENT VARIABLES



See [Dependent Variables](#) for details.

	See Settings for the Heat Transfer in Shells Interface for a description of the other settings.
	<ul style="list-style-type: none">• Handling Frames in Heat Transfer• Feature Nodes for the Heat Transfer in Shells Interface• Theory for Heat Transfer in Thin Structures
	<i>Shell Conduction</i> : 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)
- 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)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (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)
- 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)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (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.



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.

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 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)
- 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)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (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 (Its)** 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 `Its`.

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 equation defined on boundaries where surface-to-surface radiation is enabled corresponds to the radiosity method equation.

From the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Surface-to-Surface Radiation** to select physics features from the context menu. For the **Surface-to-Surface Radiation** interface, select a **Stationary** or **Time Dependent** study as a preset study type. The surface-to-surface radiation is always stationary (that is, the radiation time scale is assumed to be shorter than any other time scale), but the physics interface is compatible with all standard study types.



Absolute (thermodynamical) temperature units must be used. See [Specifying Model Equation Settings](#) in the *COMSOL Multiphysics Reference Manual*.

In this section:

- [Settings for the Surface-to-Surface Radiation Interface](#)
- [Feature Nodes for the Surface-to-Surface Radiation Interface](#)

Settings for the Surface-to-Surface Radiation Interface

The **Label** is the default physics interface name.

The **Name** is used primarily as a scope prefix for variables defined by the physics interface. Refer to such physics interface variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (`_`) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first physics interface in the model) is `rad`.

RADIATION SETTINGS

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, $[0, \lambda_{\text{sol/amb}}]$, (solar radiation) and one for large wavelengths, $[\lambda_{\text{sol/amb}}, +\infty[$, (ambient radiation). It is then possible to define the **Separation point between spectral bands** (SI unit: m), $\lambda_{\text{sol/amb}}$, to adjust the wavelength intervals corresponding to the solar and ambient radiation. The surface properties can then be defined for each spectral band. In particular it is possible to define the solar absorptivity for short wavelengths and the surface emissivity for large wavelengths.
- Select **Multiple spectral bands** and set the value of the **Right endpoint** for each spectral band in the table, to define a diffuse spectral radiation model. These values should be set in an ascending order. The value of the **Left endpoint** for the next spectral band is updated in consequence. It is then 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.

Modify the **Transparent media refractive index** if it is different from 1 and corresponds to vacuum refractive index, which is usually a good approximation for air refractive index.

Also select the **Use radiation groups** check box to enable the ability to define radiation groups, which can, in many cases, speed up the radiation calculations.

Select the **Surface-to-surface radiation method: 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.

Hemicube

Hemicube is the default method for the heat transfer interfaces. The more sophisticated and general hemicube method uses a z -buffered projection on the sides of a hemicube (with generalizations to 2D and 1D) to account for shadowing effects. Think of it as rendering digital images of the geometry in five different directions (in 3D; in 2D only three directions are needed), and counting the pixels in each mesh element to evaluate its view factor.

Its accuracy can be influenced by setting the **Radiation resolution** of the virtual snapshots. The number of z -buffer pixels on each side of the 3D hemicube equals the specified resolution squared. Thus the time required to evaluate the irradiation increases quadratically with resolution. In 2D, the number of z -buffer pixels is proportional to the resolution property, and thus the time is, as well.

For an axisymmetric geometry, G_m and F_{amb} must be evaluated in a corresponding 3D geometry obtained by revolving the 2D boundaries about the axis. COMSOL Multiphysics creates this virtual 3D geometry by revolving the 2D boundary mesh into a 3D mesh. The resolution can be controlled in the azimuthal direction by setting the number of azimuthal sectors, which is the same as the number of elements to a full revolution. Try to balance this number against the mesh resolution in the rz -plane.

Direct Area Integration

COMSOL Multiphysics evaluates the mutual irradiation between surface directly, without considering which face elements are obstructed by others. This means that shadowing effects (that is, surface elements being obstructed in nonconvex cases) are not taken into account. Elements facing away from each other are, however, excluded from the integrals.

Direct area integration is fast and accurate for simple geometries with no shadowing, or where the shadowing can be handled by manually assigning boundaries to different groups.



If shadowing is ignored, global energy is not conserved. Control the accuracy by specifying a **Radiation integration order**. Sharp angles and small gaps between surfaces may require a higher integration order for accuracy but also more time to evaluate the irradiation.

Ray Shooting

The use of a ray shooting algorithm allows to generate the view factor data for the modeling of transmission and specular reflection. Select this method to enable the [Opaque Surface \(Surface-to-Surface Radiation Interface\)](#) node (for specular reflection) and [Semitransparent Surface \(Surface-to-Surface Radiation Interface\)](#) node (for transmission).

To compute the radiation intensity on surfaces, the ray shooting algorithm emits n rays in 2D and n^2 rays in 3D where n is the value selected for **Radiation resolution**. The trajectories of these rays are computed as they 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**. During the rays trajectory computation the tiling is adapted up to a numbers of time defined by the **Maximum number of adaptations**.

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.

VIEW FACTORS UPDATE

This section is available by clicking the **Show More Options** button () and selecting **Advanced Physics Options** in the **Show More Options** dialog box.

It allows controlling the frequency of view factors computation according to a given criterion in time-dependent simulation. The view factors are updated each time the **Expression** changes by more than the **Tolerance**, instead of being updated every time step.

For example, set **Expression** to t and **Tolerance** to $1e-3[s]$ to enforce the update every $1e-3s$ of simulation time.

It can be applied to models containing a **Moving Mesh** node under **Definitions**, or when the **Symmetry for Surface-to-Surface Radiation** feature uses one or more moving symmetry planes, with time-dependent definitions.

DISCRETIZATION

Surface Radiosity

Select **Linear** (the default), **Quadratic**, **Cubic**, **Quartic** or **Quintic** to define the discretization level used for the **Surface radiosity** shape function.



- [About the Heat Transfer Interfaces](#)
 - [Feature Nodes for the Surface-to-Surface Radiation Interface](#)
 - [The Surface-to-Surface Radiation Interface](#)
 - [Theory for Surface-to-Surface Radiation](#)
-

Feature Nodes for the Surface-to-Surface Radiation Interface

This section details the nodes available with [The Surface-to-Surface Radiation Interface](#) with default settings:

- [Domain Nodes for the Surface-to-Surface Radiation Interface](#)
- [Boundary Nodes for the Surface-to-Surface Radiation Interface](#)



Some nodes are only available with some COMSOL products.

For a detailed overview of the functionality available in each product, visit <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 (), found under the **Heat Transfer>Radiation** branch (), is used to model radiative heat transfer inside participating (semitransparent media) media.



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, $[0, \lambda_{\text{sol/amb}}]$, (solar radiation) and one for large wavelengths, $[\lambda_{\text{sol/amb}}, +\infty[$, (ambient radiation). It is then possible to define the **Separation point between spectral bands** (SI unit: m), $\lambda_{\text{sol/amb}}$, 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. These values should be set in an ascending order. The value of the **Left endpoint** for the next spectral band is updated in consequence. It is then possible to provide a definition of the radiative properties for each spectral band.



The first **Left endpoint** and the last **Right endpoint** are predefined and equal to 0 and $+\infty$, respectively.

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default), or **PI 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 **PI 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 **PI 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](#)



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**.

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, $[0, \lambda_{\text{sol/amb}}]$, (solar radiation) and one for large wavelengths, $[\lambda_{\text{sol/amb}}, +\infty[$, (ambient radiation). It is then possible to define the **Separation point between spectral bands** (SI unit: m), $\lambda_{\text{sol/amb}}$, 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. These values should be set in an ascending order. The value of the **Left endpoint** for the next spectral band is updated in consequence. It is then possible to provide a definition of the radiative properties for each spectral band.



The first **Left endpoint** and the last **Right endpoint** are predefined and equal to 0 and $+\infty$, respectively.

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default), or **PI 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 **PI 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, S_2 ,

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 **PI 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)
- Radiative Source
- Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

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)
- Opaque Surface (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)
- Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Incident Intensity (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](#)



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](#)

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)
- 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 i^{th} beam (dimensionless).
- I_i is the intensity of the i^{th} 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 i^{th} 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.

ABSORBING MEDIUM

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

$$G = \sum_{i=1}^{N_V} \omega_i I_i$$

and

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^{N_V} \omega_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \quad (6-1)$$

where

- \mathbf{S}_i is the i^{th} discrete ordinate.
- I_i is the i^{th} component of the radiative intensity.
- ω_j is the i^{th} quadrature weight.

If the **Radiation discretization method** is **PI approximation**, the incident radiation G is the solution of the following equation

$$\nabla \cdot (D_{P1} \nabla G) - \kappa G = 0 \quad (6-2)$$

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 i^{th} 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	PI
Optical thickness validity	All	$\tau \gg 1$
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, \dots, 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**

$\alpha_1, \dots, \alpha_{12}$ 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](#)
- [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:



$$\rho_b C_{p,b} \omega_b (T_b - T) + Q_{\text{met}}$$

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 $(\text{m}^3/\text{s})/\text{m}^3$), 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.

	<ul style="list-style-type: none"> • Theory for Bioheat Transfer • Biological Tissue • Thermal Damage
	<p><i>Hepatic Tumor Ablation</i>: Application Library path Heat_Transfer_Module/Medical_Technology/tumor_ablation</p> <p><i>Microwave Heating of a Cancer Tumor</i>: Application Library path Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy</p> <p><i>Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis</i>: Application Library path Heat_Transfer_Module/Medical_Technology/conical_dielectric_probe</p>

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 () , 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 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.

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*.



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

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:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = Q \quad (6-3)$$

$$\mathbf{q} = -(k_{\text{eff}} \nabla T + L_v \delta_p \nabla(\phi_w p_{\text{sat}})) \quad (6-4)$$

with the following material properties, fields, and source:


- $(\rho C_p)_{\text{eff}}$ (SI unit: $\text{J}/(\text{m}^3 \cdot \text{K})$) is the effective volumetric heat capacity at constant pressure.
- k_{eff} (SI unit: $\text{W}/(\text{m} \cdot \text{K})$) 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.Tmpref`, 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.

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/pml)** 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 () , 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** ()

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:

$$k_{\text{eff}} = k_s \left(1 + \frac{b w(\phi_w)}{\rho_s} \right)$$

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** $w(\phi_w)$ 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** $C_{p,s}$ 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:

$$(\rho C_p)_{\text{eff}} = \rho_s C_{p,s} + w(\phi_w) C_{p,w}$$

where

- ρ_s (SI unit: kg/m^3) is the density of the dry solid.
- $C_{p,s}$ (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) is the specific heat capacity of the dry solid.
- $w(\phi_w)$ (SI unit: kg/m^3) is the water content given by a moisture storage function.
- $C_{p,w}$ (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) 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** $w(\phi_w)$ 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.

	<ul style="list-style-type: none">• The Heat Transfer in Building Materials Interface• The Heat and Moisture Transport Interfaces• Heat and Moisture
	See Building Material Library in the <i>COMSOL Multiphysics Reference Manual</i> 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.
	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>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.

	<p>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.</p>
	<p><i>Evaporative Cooling of Water</i>: Application Library path Heat_Transfer_Module/Phase_Change/evaporative_cooling</p>
	<p>Equivalent Thermal Conductivity Correlations</p>

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.



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:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (6-5)$$
$$\mathbf{q} = -k \nabla T$$

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the fluid density.
- C_p (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) is the fluid heat capacity at constant pressure.
- k (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) 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/m³) 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**.

FLUID MATERIAL

This section is available only when the **Local Thermal Nonequilibrium** multiphysics coupling is included in the component to model porous media. It makes it possible to define different material properties for the fluid phase when the domain material corresponds to the solid phase (porous matrix) material.

Select any material from the list to define the **Fluid material**. The default uses the **Domain material**.

HEAT CONVECTION

The default **Velocity field \mathbf{u}** is **User defined**. For **User defined** enter values or expressions for the components based on space dimensions. Or select an existing velocity field in the component (for example, **Velocity field (spf)** from a **Laminar Flow** interface). 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 \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 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 variables for heat fluxes and total energy fluxes. It is also used if the ideal gas law is applied.

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**, the **Heat capacity at constant pressure**, and the **Ratio of specific heats** are automatically taken **From material** and no further setting is required. 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 $R = 8.314 \text{ J}/(\text{mol}\cdot\text{K})$, 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.

	<ul style="list-style-type: none"> • Local Thermal Nonequilibrium • Theory for Heat Transfer in Fluids
	<ul style="list-style-type: none"> • When the density is taken from a Nonisothermal Flow coupling node, the Heat capacity at constant pressure and the Ratio of specific heats 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 <i>CFD Module User's Guide</i> for details.
	<p>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.</p>
	<p><i>Heat Sink</i>: Application Library path Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/heat_sink</p>



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

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>Fluid

Fluid (Porous Medium)

This node defines the velocity field and material properties of the mobile fluid used in [Equation 6-13](#) 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.

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.Tmpref`, 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/m³ or kg/m³) 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 \mathbf{u}** is **User defined**. For **User defined**, enter values or expressions for the components based on space dimensions. Or select an existing velocity field in the component (for example, **Velocity field (spf)** from a **Laminar Flow** interface). 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.

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 variables for heat fluxes and total energy fluxes. It is also used if the ideal gas law is applied.

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 $R = 8.314 \text{ J}/(\text{mol}\cdot\text{K})$, 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** (default) 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** (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.



LOCATION IN USER INTERFACE

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} .

The two option buttons in the same group control the distribution function $f(z_{\text{geo}})$:

- 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

$$\rho_{\text{geo}} = \frac{\sum_i \theta_{si} \rho_{si}}{\sum_i \theta_{si}}$$

- If **Porous media** is selected, it calculates the geothermal density based on all mobile and immobile components of the porous medium:

$$\rho_{\text{geo}} = \sum_i \theta_{si} \rho_{si} + \epsilon_p \rho_f + \theta_{\text{imf}} \rho_{\text{imf}}$$

- 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/>

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Porous Media>Porous Medium>Geothermal Heating

Ribbon

Physics Tab with **Porous Medium** selected in the model tree:

Attributes>Geothermal Heating

Heat Source

This node describes heat generation within the domain. You express heating and cooling with positive and negative values, respectively. Add one or more nodes as needed — all heat sources within a domain contribute to the total heat source.

The **Heat Source** node adds a source term Q to the right-hand side of the heat equation:

$$Q = Q_0$$

Specify Q_0 as the heat 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

Click the **General source** (default), **Linear source**, or **Heat rate** buttons.

- For **General source** enter a value for the distributed heat source Q_0 when the default option (**User defined**) is selected. See also [Additional General Source Options](#) to use predefined heat sources available from other interfaces.
- For **Linear source** enter a value for the **Production/absorption coefficient** q_s used in the predefined linear expression. The advantage of writing the source on this form is that it can be accounted for in the streamline diffusion stabilization. The stabilization applies when q_s is independent of the temperature, but some stability can be gained as long as q_s is only weakly dependent on the temperature.
- For **Heat rate** enter a value for the heat rate P_0 . In this case $Q_0 = P_0/V$, where V is the total volume of the selected domains.

Additional General Source Options

For the general heat source Q_0 there are predefined heat sources available (in addition to a **User defined** heat source) when simulating heat transfer together with electrical or electromagnetic interfaces. Such sources represent, for example, ohmic heating and induction heating. Depending on additional physics interfaces, the following are available:




- With the addition of an Electric Currents interface, the **Total power dissipation density (ec)** heat source is available from the **General source** list.
- With the addition of any version of the Electromagnetic Waves interface (which requires the RF Module), the **Total power dissipation density (emw)** and **Electromagnetic power loss density (emw)** heat sources are available from the **General source** list.
- With the addition of a Magnetic Fields interface (a 3D component requires the AC/DC Module), the **Electromagnetic heating (mf)** heat source is available from the **General source** list.
- With the addition of a Magnetic and Electric Fields interface (which requires the AC/DC Module), the **Electromagnetic heating (mef)** heat source is available from the **General source** list.
- For the Heat Transfer in Porous Media interface, with the addition of interfaces from the 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.

	<p>For the definition of a localized heat source, see Line Heat Source, Line Heat Source and Point Heat Source.</p> <p>For the definition of a heat on a boundary, see Boundary Heat Source.</p>
	<p><i>Forced Convection Cooling of an Enclosure with Fan and Grille:</i> Application Library path Heat_Transfer_Module/ Power_Electronics_and_Electronic_Cooling/electronic_enclosure_cooling</p>
	<p>For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/</p>

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Heat Source

Heat Transfer in Solids>Heat Source

Heat Transfer in Fluids>Heat Source

Heat Transfer in Porous Media>Heat Source

Heat Transfer in Building Materials>Heat Source

Bioheat Transfer>Heat Source

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>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.

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

$$\theta_{\text{imf}} = \sum_i \theta_{\text{imfi}}$$

The volume fraction available for mobile fluids (that is, the effective porosity) is then calculated from

$$\varepsilon_p = 1 - \theta_s - \theta_{\text{imf}}$$

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

$$k_{\text{imf}} = \sum_i \theta_{\text{imfi}} k_{\text{imfi}}$$

In the **Porous Medium** node, the effective thermal conductivity is defined as:

- **Volume average** (default), which calculates the effective conductivity of the solid-fluid system as the weighted arithmetic mean of the conductivities of the fluids and the porous matrix:

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

- **Reciprocal average**, which calculates the effective conductivity of the solid-fluid system as the weighted harmonic mean of the conductivities of the fluids and the porous matrix:

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

- **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:

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

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** $C_{p, \text{imf}}$ 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:

$$\rho_{\text{imf}} = \sum_i \theta_{\text{imfi}} \rho_{\text{imfi}}$$

$$C_{p, \text{imf}} = \frac{\sum_i \theta_{\text{imfi}} \rho_{\text{imfi}} C_{p, \text{imfi}}}{\sum_i \theta_{\text{imfi}} \rho_{\text{imfi}}}$$

In the **Porous Medium** node, the effective volumetric heat capacity at constant pressure is defined as:

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



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.

INITIAL VALUES

For **User defined**, enter a value or expression for the initial value of the **Temperature T** (SI unit: K). The default value is approximately room temperature, 293.15 K (20°C). Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Initial Values

Heat Transfer in Solids>Initial Values

Heat Transfer in Fluids>Initial Values

Heat Transfer in Porous Media>Initial Values

Heat Transfer in Building Materials>Initial Values

Bioheat Transfer>Initial Values

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>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 **PI 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.

PI Approximation

When **PI 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 $G = 4\pi I_b(T)$. 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 $G = 4\pi I_b(T)$. 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

Ribbon

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 6-17](#) 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 6-17](#) 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 6-17](#) 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.

THERMODYNAMICS

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** (). 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.

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 4-52](#).
- 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.



It is not possible to couple an **Isothermal Domain** with the **Laminar Flow** interface through the **Nonisothermal Flow** multiphysics condition.



- The following conditions of heat exchange can be applied at isothermal domain interfaces: thermal insulation, continuity, ventilation, convective heat flux, and thermal contact. See [Isothermal Domain Interface](#) for more details.
- Also see [Theory for Lumped Isothermal Domain](#).



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}_1 and the saturation s_1 of the liquid water phase used in Equation 6-6 of the **Moist Porous Medium** parent node. The density ρ_1 (SI unit: kg/m^3), the heat capacity at constant pressure $C_{p,1}$ (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$), and the thermal conductivity k_1 (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) are defined as functions of temperature.

LIQUID WATER PROPERTIES

The **Liquid water saturation**, s_1 , 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}_1 , 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** ()



- [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: $\text{J}/(\text{kg}\cdot\text{K})$), and thermal conductivity k (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) 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** (). 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).


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**.

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** \mathbf{u} is **User defined**. For **User defined** enter values or expressions for the components based on space dimensions. Or select an existing velocity field in the component (for example, **Velocity field (spf)** from a **Laminar Flow** interface). 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.

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_{ϕ_w} and **Relative humidity, absolute pressure condition** p_{ϕ_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.



	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.
	<i>Evaporative Cooling of Water</i> : Application Library path Heat_Transfer_Module/Phase_Change/evaporative_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>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

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>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: $\text{J}/(\text{kg}\cdot\text{K})$), and the thermal conductivity k_g (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) 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:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + (\rho_g C_{p,g} \mathbf{u}_g + \rho_l C_{p,l} \mathbf{u}_l) \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + Q_{\text{evap}} \quad (6-6)$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \quad (6-7)$$

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

$$(\rho C_p)_{\text{eff}} = \theta_s \rho_s C_{p,s} + \epsilon_p [(1 - s_l) \rho_g C_{p,g} + s_l \rho_l C_{p,l}]$$

where s_l 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:

$$(\rho C_p)_{\text{eff}} = \sum_i \theta_{si} \rho_{si} C_{p,si} + \epsilon_p [(1 - s_l) \rho_g C_{p,g} + s_l \rho_l C_{p,l}]$$

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, as the 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.

EFFECTIVE THERMAL CONDUCTIVITY

This section defines the averaging model for the computation of the **Effective conductivity** k_{eff} , taking into account the properties of the solid matrix, moist air, and liquid water. The following options are available:

- **Volume average** (default), which calculates the effective conductivity of the moist porous medium as the weighted arithmetic mean of the conductivities of the porous matrix, moist air, and liquid water:

$$k_{\text{eff}} = \theta_s k_s + \varepsilon_p [(1 - s_1)k_g + s_1 k_l]$$

- **Reciprocal average**, which calculates the effective conductivity of the moist porous medium as the weighted harmonic mean of the conductivities of the porous matrix, moist air, and liquid water:

$$\frac{1}{k_{\text{eff}}} = \frac{\theta_s}{k_s} + \frac{\varepsilon_p}{(1 - s_1)k_g + s_1 k_l}$$

- **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:

$$k_{\text{eff}} = k_s^{\theta_s} \cdot [(1 - s_1)k_g + s_1 k_l]^{\varepsilon_p}$$

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:

- **Volume average:**

$$k_{\text{eff}} = \sum_i \theta_{si} k_{si} + \varepsilon_p [(1 - s_1)k_g + s_1 k_l]$$

- **Reciprocal average:**

$$\frac{1}{k_{\text{eff}}} = \sum_i \theta_{si} \frac{\sum_i \theta_{si}}{\sum_i \theta_{si} k_{si}} + \frac{\varepsilon_p}{(1-s_1)k_g + s_1 k_l}$$

- **Power law:**

$$k_{\text{eff}} = \left(\frac{\sum_i \theta_{si} k_{si}}{\sum_i \theta_{si}} \right)^{\sum_i \theta_{si}} \cdot [(1-s_1)k_g + s_1 k_l]^{\varepsilon_p}$$

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 **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 **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 **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.



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



$$k_R = \frac{16n_r^2\sigma T^3}{3\beta_R}$$

where n_r is the refractive index (dimensionless), σ is the Stefan-Boltzmann constant (SI unit: $\text{W}/(\text{m}^2\cdot\text{K}^4)$), 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** (). 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

LOCATION IN USER INTERFACE

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 $q_{0,u}$ for the upside heat flux and a heat flux $q_{0,d}$ 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](#):

$$d_z \rho C_p \frac{\partial T}{\partial t} - \nabla \cdot \mathbf{q} = d_z Q + q_0 \quad (6-8)$$

$$d_z \rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla T \right) = \nabla \cdot \mathbf{q} + d_z Q + q_0 \quad (6-9)$$

$$q_0 = q_{0,u} + q_{0,d}$$

The convective heat flux adds the following contribution

$$q_0 = h_u (T_{\text{ext},u} - T) + h_d (T_{\text{ext},d} - T)$$

UPSIDE INWARD HEAT FLUX

The available options are **General inward heat flux** and **Convective heat flux**. The settings are the same as for the [Heat Flux](#) node.

DOWNSIDE INWARD HEAT FLUX

The available options are **General inward heat flux** and **Convective heat flux**. The settings are the same as for the [Heat Flux](#) node.



See [Out-of-Plane Heat Transfer](#) for the formulation of out-of-plane heat transfer in 1D, 1D axisymmetric, and 2D geometries.

See also [Out-of-Plane Domain Fluxes](#).



Upside and downside settings can be visualized by plotting the global normal vector (nx, ny, nz), that always points from downside to upside. Note that the normal vector (ht.nx, ht.ny, ht.nz) may be oriented differently.

See [Tangent and Normal Variables](#) in the *COMSOL Multiphysics Reference Manual*.



Out-of-Plane Heat Transfer for a Thin Plate: Application Library path `Heat_Transfer_Module/Verification_Examples/thin_plate`

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Out-of-Plane>Out-of-Plane Heat Flux

Heat Transfer in Solids>Out-of-Plane>Out-of-Plane Heat Flux

Heat Transfer in Fluids>Out-of-Plane>Out-of-Plane Heat Flux
Heat Transfer in Porous Media>Out-of-Plane>Out-of-Plane Heat Flux
Heat Transfer in Building Materials>Out-of-Plane>Out-of-Plane Heat Flux
Bioheat Transfer>Out-of-Plane>Out-of-Plane Heat Flux

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>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](#):

$$\varepsilon_u \sigma (T_{\text{amb, u}}^4 - T^4) + \varepsilon_d \sigma (T_{\text{amb, d}}^4 - T^4)$$

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** $T_{\text{amb, u}}$. 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**.

DOWNSIDE PARAMETERS

Follow the instructions for the **Upside Parameters** section to define the downside parameters ϵ_d and $T_{\text{amb}, d}$.

	Out-of-Plane Heat Transfer
---	----------------------------

	<p>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.</p> <p>See Tangent and Normal Variables in the <i>COMSOL Multiphysics Reference Manual</i>.</p>
---	---

	<p><i>Out-of-Plane Heat Transfer for a Thin Plate</i>: Application Library path Heat_Transfer_Module/Verification_Examples/thin_plate</p>
---	--

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Out-of-Plane>Out-of-Plane Radiation

Heat Transfer in Solids>Out-of-Plane>Out-of-Plane Radiation

Heat Transfer in Fluids>Out-of-Plane>Out-of-Plane Radiation

Heat Transfer in Porous Media>Out-of-Plane>Out-of-Plane Radiation

Heat Transfer in Building Materials>Out-of-Plane>Out-of-Plane Radiation

Bioheat Transfer>Out-of-Plane>Out-of-Plane Radiation

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>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: $W/(m^2 \cdot sr)$), 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)

G is 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*

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega'$$

- $\beta = \kappa + \sigma_s$ 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

$$G = \sum_{i=1}^{LV} \omega_i I_i \quad (6-10)$$

and

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^{LV} \omega_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \quad (6-11)$$

where

- \mathbf{S}_i is the i th discrete ordinate.
- I_i is the i th component of the radiative intensity.
- ω_j is the i th quadrature weight.

If the **Radiation discretization method** is **PI approximation**, G is the solution of the following equation

$$\nabla \cdot (D_{P1} \nabla G) - \kappa(G - 4\pi I_b) = 0 \quad (6-12)$$

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 i^{th} 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	PI
Optical thickness validity	All	$\tau \gg 1$
Absorption modeling	Yes	Yes
Emission modeling	Yes	Yes

TABLE 6-3: DISCRETIZATION METHODS FOR PARTICIPATING MEDIUM (RPM INTERFACE)

OPTION	DOM	PI
Scattering modeling	Linear	Linear
	Polynomial	
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 () , 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, \dots, 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:

$$K = \frac{1}{4\pi} \cdot \int \frac{1 - \eta^2}{4\pi(1 + \eta^2 - 2\eta\mu_0)^{3/2}} d\Omega$$



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**

$\alpha_1, \dots, \alpha_{12}$ or the **Anisotropy parameter** μ to obtain piecewise constant coefficients on each spectral band.

	<ul style="list-style-type: none"> • Theory for Radiation in Participating Media • Discrete Ordinates Method (DOM) • P1 Approximation Theory
	<p><i>Radiative Heat Transfer in a Utility Boiler</i>: Application Library path Heat_Transfer_Module/Thermal_Radiation/boiler</p> <p><i>Radiative Cooling of a Glass Plate</i>: Application Library path Heat_Transfer_Module/Thermal_Radiation/glass_plate</p> <p><i>Radiative Heat Transfer in Finite Cylindrical Media</i>: Application Library path Heat_Transfer_Module/Verification_Examples/cylinder_participating_media</p> <p><i>Radiative Heat Transfer in Finite Cylindrical Media — P1 Method</i>: Application Library path Heat_Transfer_Module/Verification_Examples/cylinder_participating_media_p1</p>

LOCATION IN USER INTERFACE

Context Menus

Radiation in Participating Media>Participating Medium

Ribbon


Physics Tab with **Radiation in Participating Media** selected:

Domains>Radiation in Participating Media>Participating Medium

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 $T_{pc,j \rightarrow j+1} - \Delta T_{j \rightarrow j+1}/2$ and $T_{pc,j \rightarrow j+1} + \Delta T_{j \rightarrow j+1}/2$, releasing a total heat per unit volume equal to $L_{j \rightarrow j+1}$.

The **Phase change temperature between phase 1 and phase 2** $T_{pc,1 \rightarrow 2}$ 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** $\Delta T_{1 \rightarrow 2}$ 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 $\Delta T_{j \rightarrow j+1}$ 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** $L_{1 \rightarrow 2}$ 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 $L_{j \rightarrow j+1}$ 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 $T < T_{pc, 1 \rightarrow 2}$, while the material properties of phase 2 hold for $T > T_{pc, 1 \rightarrow 2}$.

When more than one transition is modeled, the number of phases exceeds 2, and new variables are created (for example, $T_{pc, 2 \rightarrow 3}$, $\Delta T_{2 \rightarrow 3}$ or $L_{2 \rightarrow 3}$). The phase change temperatures $T_{pc, j \rightarrow j+1}$ are increasing and satisfy

$$T_{pc, 1 \rightarrow 2} < T_{pc, 2 \rightarrow 3} < \dots$$

This defines distinct domains of temperature bounded by $T_{pc, j-1 \rightarrow j}$ and $T_{pc, j \rightarrow j+1}$ where the material properties of phase j only apply.

In addition, the values of $\Delta T_{j \rightarrow j+1}$ are chosen so that the ranges between $T_{pc, j \rightarrow j+1} - \Delta T_{j \rightarrow j+1}/2$ and $T_{pc, j \rightarrow j+1} + \Delta T_{j \rightarrow j+1}/2$ do not overlap. If this condition is not satisfied, unexpected behavior can occur because some phases would never form completely. The values of $\Delta T_{j \rightarrow j+1}$ 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. The default is 1 W/(m·K).
- **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/m³.
- **Heat capacity at constant pressure** $C_{p,i}$. The default is 4200 J/(kg·K).
- **Ratio of specific heats** γ_i . The default is 1.1.

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

Cooling and Solidification of Metal: Application Library path **Heat_Transfer_Module/Thermal_Processing/cooling_solidification_metal**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids>Solid>Phase Change Material

Heat Transfer in Fluids>Fluid>Phase Change Material

Heat Transfer in Porous Media>Porous Medium>Phase Change Material

More locations are available. For example:

Heat Transfer in Fluids>Solid>Phase Change Material

Ribbon

Physics Tab with **Solid**, **Fluid**, or **Porous Medium** 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 [Equation 6-6](#) or [Equation 6-13](#) 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:

$$\epsilon_p = 1 - \sum_i \theta_{si} - \sum_i \theta_{imfi}$$

where θ_{si} and θ_{imfi} are the porosities 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:

$$k_s = \frac{k_b}{\theta_s}$$

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:

$$\rho_s = \sum_i \theta_{si} \rho_{si}$$

$$C_{p, s} = \frac{\sum_i \theta_{si} \rho_{si} C_{p, si}}{\sum_i \theta_{si} \rho_{si}}$$

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

$$C_{p, s} = \frac{C_{p, b}}{\theta_s}$$

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 uses the following version of the heat equation 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:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (6-13)$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \quad (6-14)$$

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the density of the mobile fluid.
- C_p (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) is the heat capacity at constant pressure of the mobile fluid.
- $(\rho C_p)_{\text{eff}}$ (SI unit: $\text{J}/(\text{m}^3\cdot\text{K})$) is the effective volumetric heat capacity at constant pressure, defined by an averaging model to take into account both solid matrix and fluid properties.
- \mathbf{q} is the conductive heat flux (SI unit: W/m^2).
- \mathbf{u} (SI unit: m/s) is the velocity field of the mobile fluid, either an analytic expression or the velocity field from a Fluid Flow interface. \mathbf{u} should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area. The average linear velocity (the velocity within the pores) can be calculated as $\mathbf{u}_L = \mathbf{u}/\epsilon_p$, where ϵ_p is the porosity.
- k_{eff} (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) is the effective thermal conductivity (a scalar or a tensor if the thermal conductivity is anisotropic), defined by an averaging model to take into account both solid matrix and fluid properties.
- 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 [Viscous Dissipation](#) subnode for example.

For a steady-state problem, the temperature does not change with time and the first term disappears.

When no immobile fluid is present in the pore space, the effective volumetric heat capacity at constant pressure is defined as

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

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.

EFFECTIVE THERMAL CONDUCTIVITY

This section defines the averaging model for the computation of the **Effective conductivity** taking into account the properties of the solid matrix and the mobile fluid. The following options are available:

- **Volume average** (default), which calculates the effective conductivity of the solid-fluid system as the weighted arithmetic mean of the conductivities of the fluid and the porous matrix:

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

- **Reciprocal average**, which calculates the effective conductivity of the solid-fluid system as the weighted harmonic mean 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:

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

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:

- **Volume average:**

$$k_{\text{eff}} = \sum_i \theta_{si} k_{si} + \varepsilon_p k_f$$

- **Reciprocal average:**

$$\frac{1}{k_{\text{eff}}} = \sum_i \theta_{si} \frac{\sum_i \theta_{si}}{\sum_i \theta_{si} k_{si}} + \frac{\varepsilon_p}{k_f}$$

- **Power law:**

$$k_{\text{eff}} = \left(\frac{\sum_i \theta_{si} k_{si}}{\sum_i \theta_{si}} \right)^{\sum_i \theta_{si}} \cdot k_f^{\varepsilon_p}$$



- See [Immobile Fluids \(Porous Medium\)](#) to take into account the presence of immobile fluids in the expression of effective thermal conductivity.
- 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 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 Porous Media](#)
- [Local Thermal Nonequilibrium](#)



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 in Fluids** equation to model the result of heating under adiabatic compression as well as some thermoacoustic effects:

$$Q_p = \alpha_p T \left(\frac{\partial p_A}{\partial t} + \mathbf{u} \cdot \nabla p_A \right) \quad (6-15)$$

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

More locations are available. For example:

Heat Transfer in Solids>Fluid>Pressure Work

Heat Transfer in Moist Air>Moist Air>Pressure Work

Ribbon

Physics Tab with **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 **PI 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 $Q_0 = P_0V$, 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:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (6-16)$$
$$\mathbf{q} = -k \nabla T$$

with the following material properties, fields, and sources:


- ρ (SI unit: kg/m^3) is the alloy's density.
- C_p (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) is the alloy's heat capacity at constant pressure.
- k (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) 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:

$$k = \xi k_M + (1 - \xi) k_A$$

- **Reciprocal average**, which calculates the effective conductivity of the alloy as the weighted harmonic mean of austenite and martensite conductivities:

$$\frac{1}{k} = \frac{\xi}{k_M} + \frac{1 - \xi}{k_A}$$

- **Power law**, which calculates the effective conductivity of the alloy as the weighted geometric mean of austenite and martensite conductivities:

$$k = k_M^\xi \cdot k_A^{(1 - \xi)}$$

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** $C_{p,A}$ 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** $C_{p,M}$ is taken **From material**. For **User defined** enter a value or expression.

LOCATION IN USER INTERFACE

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:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (6-17)$$
$$\mathbf{q} = -k \nabla T$$

with the following material properties, fields, and sources:


- ρ (SI unit: kg/m^3) is the solid density.
- C_p (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) is the solid heat capacity at constant pressure.
- k (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) 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 `m.input.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.

SOLID MATERIAL

This section is available only when the **Local Thermal Nonequilibrium** multiphysics coupling is included in the component to model porous media. It makes it possible to define different material properties for the porous matrix and the fluid.

Select any material from the list to define the **Solid material**. The default uses the **Domain material**. See [Material Density in Features Defined in the Material Frame](#) for the setting of a temperature-dependent density.

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} .



Fourier's law assumes that the thermal conductivity tensor is symmetric. A nonsymmetric tensor can lead to unphysical results.

THERMODYNAMICS, SOLID

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.alphaTdx, ht.alphaTdy, and so on (using the default physics name ht). The single scalar mean thermal diffusivity ht.alphaTMean is the mean value of the diagonal elements α_{xx} , α_{yy} , and α_{zz} . The denominator ρC_p is the effective volumetric heat capacity which is also available as a predefined quantity, ht.C_eff.



- [Local Thermal Nonequilibrium](#)
- [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:

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

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** $T_{d,c}$ to define the (low) temperature that the tissue needs to reach to start getting damaged. The default is 273.15 K.
- **Damage time** $t_{d,c}$ to define the time needed for the necrosis to happen while the temperature is below $T_{d,c}$. The default is 50 s.
- **Necrosis temperature** $T_{n,c}$ to define the (low) temperature to be reached for the necrosis to happen instantly. The default is 253.15 K.
- **Enthalpy change** $L_{d,c}$ 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:

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

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.

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 $\nabla \cdot (k_{\text{disp}} \nabla T)$ 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} :

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T = \nabla \cdot (k_{\text{eff}} \nabla T) + Q$$

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:

$$(k_{\text{disp}})_{ij} = \rho C_p D_{ij}$$

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

$$\lambda_{ijkl} = \lambda_{\text{tr}} \delta_{ij} \delta_{kl} + \frac{\lambda_{\text{lo}} - \lambda_{\text{tr}}}{2} (\delta_{ik} \delta_{jl} + \delta_{il} \delta_{jk})$$

When the **Horizontal and vertical transverse dispersivities** option is selected in the **Define transverse dispersivity from** list, the **Transverse horizontal dispersivity** $\lambda_{\text{tr}, \text{h}}$ and **Transverse vertical dispersivity** $\lambda_{\text{tr}, \text{v}}$ 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

$$D_{ij} = \frac{1}{|\mathbf{u}|} \begin{bmatrix} \lambda_{10}u^2 + \lambda_{tr,h}v^2 + \lambda_{tr,v}w^2 & (\lambda_{10} - \lambda_{tr,h})uv & (\lambda_{10} - \lambda_{tr,v})uw \\ (\lambda_{10} - \lambda_{tr,h})uv & \lambda_{tr,h}u^2 + \lambda_{10}v^2 + \lambda_{tr,v}w^2 & (\lambda_{10} - \lambda_{tr,v})vw \\ (\lambda_{10} - \lambda_{tr,v})uw & (\lambda_{10} - \lambda_{tr,v})vw & \lambda_{tr,v}u^2 + \lambda_{tr,v}v^2 + \lambda_{10}w^2 \end{bmatrix}$$



The former formulation corresponds to the general formulation when $\lambda_{tr,h} = \lambda_{tr,v}$.



Porous Medium



The **Thermal Dispersion** node is only available with some COMSOL products. 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>Thermal Dispersion

Ribbon

Physics Tab with **Porous Medium** selected in the model tree:

Attributes>Thermal Dispersion

Thermoelastic Damping

This subnode should be used to model heat generation due to changes in stress, which may be important in small structures vibrating at high frequencies.

THERMOELASTIC DAMPING

The **Thermoelastic damping** Q_{ted} should be specified either as a **User defined** value, or as the thermoelastic damping contribution straight from the solid mechanics interfaces, when you add a **Thermal Expansion** subnode (with the Structural Mechanics Module). In the latter case it is defined by

$$Q_{\text{ted}} = -\alpha T : \frac{\partial S}{\partial t}$$

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*

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids>Solid>Thermoelastic Damping

More locations are available. For example:

Heat Transfer in Fluids>Solid>Thermoelastic Damping

Ribbon

Physics Tab with **Solid** selected in the model tree:

Attributes>Thermoelastic Damping

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.



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

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>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 6-17](#), 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
---	-------

	<i>Heat Generation in a Disc Brake</i> : Application Library path Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc
	<i>Friction Stir Welding of an Aluminum Plate</i> : 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

More locations are available. For example:

Heat Transfer in Solids>Fluid>Viscous Dissipation

Heat Transfer in Solids>Porous Medium>Viscous Dissipation

Heat Transfer in Moist Air>Moist Air>Viscous Dissipation

Ribbon

Physics Tab with **Fluid**, **Moist Air**, or **Porous Medium** 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
- 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)



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 Interface)
- 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 **Pair Boundary Heat Source**, it also prescribes that the temperature field is continuous across the pair. Note that in this case the source term is applied on the source side.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must 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

Click the **General source** (default) or **Heat rate** button.

- For **General source** enter a value for the boundary heat source Q_b when the default option, **User defined**, is selected.

A positive Q_b corresponds to heating and a negative Q_b corresponds to cooling. For the general boundary heat source Q_b , there are predefined heat sources available when simulating heat transfer together with electrical or electromagnetic interfaces. Such sources represent, for example, ohmic heating and induction heating.

- For **Heat rate** enter the heat rate P_b . In this case $Q_b = P_b/A$, where A is the total area of the selected boundaries.

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.

	<p>Upside and downside settings can be visualized by plotting the global normal vector (n_x, n_y, n_z), that always points from downside to upside. Note that the normal vector $(ht.nx, ht.ny, ht.nz)$ may be oriented differently.</p> <p>See Tangent and Normal Variables in the <i>COMSOL Multiphysics Reference Manual</i>.</p>
	<ul style="list-style-type: none"> • Handling Frames in Heat Transfer • About the Heat Transfer Interfaces
	<p><i>Evaporative Cooling of Water</i>: Application Library path Heat_Transfer_Module/Phase_Change/evaporative_cooling</p>
	<p>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.</p> <p>See Identity and Contact Pairs in the <i>COMSOL Multiphysics Reference Manual</i> for more details.</p>

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Heat Sources>Boundary Heat Source

Heat Transfer in Solids>Heat Sources>Boundary Heat Source

Heat Transfer in Fluids>Heat Sources>Boundary Heat Source

Heat Transfer in Porous Media>Heat Sources>Boundary Heat Source

Heat Transfer in Building Materials>Heat Sources>Boundary Heat Source

Bioheat Transfer>Heat Sources>Boundary Heat Source

Heat Transfer in Solids>Pairs>Pair Boundary Heat Source

Heat Transfer in Fluids>Pairs>Pair Boundary Heat Source

Heat Transfer in Porous Media>Pairs>Pair Boundary Heat Source

Heat Transfer in Building Materials>Pairs>Pair Boundary Heat Source

Bioheat Transfer>Pairs>Pair Boundary Heat Source

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*>**Boundary Heat Source**

Pairs>*interface*>**Pair Boundary Heat Source**

Continuity

This node can be added to pairs. It prescribes that the temperature field is continuous across the pair. **Continuity** is only suitable for pairs where the boundaries match.

PAIR SELECTION

Choose the pair to apply this condition to. A pair must 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 Solids>**Pairs**>**Continuity**

Heat Transfer in Fluids>**Pairs**>**Continuity**

Heat Transfer in Porous Media>**Pairs**>**Continuity**

Bioheat Transfer>**Pairs**>**Continuity**

Heat Transfer in Shells>**Pairs**>**Continuity**

Surface-to-Surface Radiation>**Pairs**>**Continuity**

Radiation in Participating Media>**Pairs**>**Continuity**

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**, **Bioheat Transfer**, **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 **PI 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 **PI approximation**, this boundary condition is not available since it is not needed.



Radiative Heat Transfer in a Utility Boiler: Application Library path
Heat_Transfer_Module/Thermal_Radiation/boiler

LOCATION IN USER INTERFACE

Context Menus

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** \mathbf{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 **Deposited beam power density**, Q_b (SI unit: W/m^2).

For **Built-in beam profiles**, enter a value for the **Deposited beam power** P_0 and the coordinates of the **Beam origin point** \mathbf{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.

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.



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 Solids>Heat Sources>Deposited Beam Power

Heat Transfer in Fluids>Heat Sources>Deposited Beam Power

Heat Transfer in Porous Media>Heat Sources>Deposited Beam Power

Bioheat Transfer>Heat Sources>Deposited Beam Power

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>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.

LOCATION IN USER INTERFACE

Context Menus

Surface-to-Surface Radiation>Diffuse Mirror

Ribbon

Physics Tab with **Surface-to-Surface Radiation** selected:

Boundaries>Surface-to-Surface Radiation>Diffuse Mirror

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 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 a **Radiation direction** based on the geometric normal (n_x , n_y , n_z): **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.
- 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.
- 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.
- Select **Both sides** if the surface radiates on both sides.

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 a **Radiation direction** for each spectral band: **Opacity controlled** (default), **Negative normal direction**, **Positive normal direction**, **Both sides**, or **None**. The **Radiation direction** defines the radiation direction for each spectral band similarly as when **Wavelength dependence of 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.

None is used when adjacent domains are either both transparent or both opaque for a given spectral band.



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

Ambient Temperature

The ambient temperature T_{amb} should be set for the computation of $e_b(T_{\text{amb}})$, the ambient blackbody hemispherical total emissive power, used for the evaluation of the ambient irradiation $G_{\text{amb}}=F_{\text{amb}}\epsilon_{\text{amb}}e_b(T_{\text{amb}})$.

Select the **Define ambient temperature on each side** check box when the ambient temperature differs between the sides of a boundary. This is needed to define ambient temperature for a surface that radiates on both sides and that is exposed to a hot temperature on one side (for example, fire) and to a cold temperature on the other side (for example, external temperature). By default, **Define ambient temperature on each side** is not selected.

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 temperature on each side** is selected, define the **Ambient temperature, upside** $T_{\text{amb}, \text{u}}$ and **Ambient temperature, downside** $T_{\text{amb}, \text{d}}$, 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 $G_{\text{amb}}=F_{\text{amb}}\epsilon_{\text{amb}}e_b(T_{\text{amb}})$.

Select the **Define ambient emissivity on each side** check box when the ambient emissivity differs between the sides of a boundary. In this case the ambient emissivity user inputs are duplicated to be defined for each side.

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 $G_{\text{amb},i}$ 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 $\epsilon_{\text{amb},i}$. If the average value of the emissivity on each band $\epsilon_{\text{amb},i}$ 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** $\epsilon_{\text{amb},i}$ for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength.

When **Define ambient emissivity on each side** is selected, define the **Ambient emissivity, upside** $\epsilon_{\text{amb,u}}$ and **Ambient emissivity, downside** $\epsilon_{\text{amb,d}}$ respectively. The geometric normal points from the downside to the upside.

Diffuse Irradiance

By default, a diffuse irradiation contribution I_{diff} is included into the external irradiation. For **User defined**, enter a value or expression. When considering solar irradiation, it takes into account the irradiation from the sun, scattered by the atmosphere, and assumed to be isotropic. Else, select a **Clear sky noon diffuse horizontal irradiance** defined in an **Ambient Properties** node under **Definitions**.

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 **Downside** 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 **Surface emissivity** ε (dimensionless) uses values **From material**. This is a property of the material surface that depends both on the material itself and the structure of the surface. Make sure that a material is defined at the boundary level (by default materials are defined at the domain level).
- For **User defined**, set a value or expression. You can define a temperature-dependent emissivity using the variable `rad.T`.

Select the **Define surface emissivity on each side** check box to set specific values on each side. The **Surface emissivity, upside** and **Surface emissivity, downside** should be set.





If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Surface emissivity** ε (dimensionless) uses values **From material**.
- When **Surface emissivity** is set to **User defined**, enter a value or expression for the **Surface 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 **Surface emissivity** is set to **User defined for each band**, enter a value for the **Surface 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 initial surface emissivity 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.

When the **Radiation direction** is set to **None** for a spectral band, the information set for this spectral band in the **Surface Emissivity** section is not used.

	In the notation used here, B_i stands for B_1, B_2, \dots 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 (n_x , n_y , n_z), that always points from downside to upside. Note that the normal vector ($ht.n_x$, $ht.n_y$, $ht.n_z$) 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.

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 \(Heat Transfer in Shells Interface\)](#)
- [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.

BOUNDARY SELECTION

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, s1mat1), 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 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.

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.Tref , 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**: **Volume average** (default) or **Power law**.

THERMODYNAMICS, POROUS MATRIX

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

Click to select the **General inward heat flux** (default), **Convective heat flux**, or **Heat rate** button.

General Inward Heat Flux

It adds q_0 to the total flux across the selected boundaries. Enter a value for q_0 to represent a heat flux that enters the domain. For example, any electric heater is well represented by this condition, and its geometry can be omitted.

Convective Heat Flux

The default option is to enter a **User defined** value for the **Heat transfer coefficient** h .

In addition, the following options are also available to control the type of convective heat flux to model: **External natural convection**, **Internal natural convection**, **External forced convection**, or **Internal forced convection**.

For all options except **User defined**, select a **Fluid: Air** (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**, $\phi_{w, \text{ext}}$, 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.



- The detailed definition of the predefined heat transfer coefficients is given in [The Heat Transfer Coefficients](#).
- For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in [Ref. 20](#).



- [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 Solids>Heat Flux

Heat Transfer in Fluids>Heat Flux

Heat Transfer in Porous Media>Heat Flux
Bioheat Transfer>Heat Flux

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>Heat Flux

Heat Source (Heat Transfer in Shells Interface)

This node describes heat generation within the shell through a $d_s Q$ contribution to the right-hand side of Equation 4-70. 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.



- 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.

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:

Boundaries>Heat Transfer in Shells>Heat Source

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 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.

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 **Boundary 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 **Boundary radiation intensity** is **User defined**, set a value or expression for I_{wall} . Alternatively, choose **Blackbody radiation** or **Graybody radiation** to define I_i as $I_b(T)$ or $\epsilon I_b(T)$, respectively (available with the Radiation in Participating Media interface only). For **Graybody radiation**, the **Surface emissivity** ϵ should be defined. If it 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 **Surface emissivity**, enter another value for ϵ . If the **Define surface emissivity on each side** check box is selected, define the **Surface emissivity, upside** ϵ_u and the **Surface emissivity, downside** ϵ_d .

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the **Boundary 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 **Boundary 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 $I_b(T)$, the fractional emissive power for each spectral band, FEP_k , and the emissivity, ϵ_k . For **Graybody radiation**, the **Surface emissivity** ϵ_k should be defined. 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 surface emissivity on each side** check box is selected, define the **Surface emissivity, upside** ϵ_u and the **Surface emissivity, downside** ϵ_d .

If **Boundary radiation intensity** is **User defined for each band**, set a value or expression for $I_{wall,k}$ 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 **Boundary radiation intensity** is **User defined distribution**, set a value or expression for $I_{\lambda,wall}$. 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 $I_{\lambda,wall,u}$ and $I_{\lambda,wall,d}$.



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](#)

LOCATION IN USER INTERFACE

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.

BEAM PROFILE

Either set the **Beam profile** as **User defined** by entering a value for the **Deposited beam power density**, I_0 (SI unit: W/m^2), or select an option among the **Built-in beam profiles**.

The **Distribution type** list provides the following options: **Gaussian** (default) or **Top-hat disk**.

Enter a value for the **Deposited beam power** P_0 (SI unit: W) and the **Beam origin point** **O** (SI unit: m).

- For **Gaussian**, enter the **Standard deviation** σ (SI unit: m).
- For **Top-hat disk**, enter the **Beam radius** R (SI unit: m). Smoothing can be applied by entering a positive **Size of transition zone** ΔR (SI unit: m). 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](#)

LOCATION IN USER INTERFACE

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:

$$-\mathbf{n} \cdot \mathbf{q} = \rho \Delta H \mathbf{u} \cdot \mathbf{n} \quad (6-18)$$

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**.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Flow Conditions>Inflow

Heat Transfer in Solids>Flow Conditions>Inflow

Heat Transfer in Fluids>Flow Conditions>Inflow

Heat Transfer in Porous Media>Flow Conditions>Inflow

Bioheat Transfer>Flow Conditions>Inflow

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>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, s1mat1), 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 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.

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**.



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.lambd` 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.



LOCATION IN USER INTERFACE

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:

- $\varphi_{d \rightarrow u}$ is the mass flux in the positive direction and $\varphi_{u \rightarrow d}$ is the mass flux in the negative direction.
- $\Phi_{d \rightarrow u}$ is the mass flow rate in the positive direction and $\Phi_{u \rightarrow d}$ 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.

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: $\text{K}\cdot\text{m}^2/\text{W}$), 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.

	<ul style="list-style-type: none"> • Isothermal Domain • Theory for Lumped Isothermal Domain • The detailed definition of the predefined heat transfer coefficients is given in The Heat Transfer Coefficients.
	<ul style="list-style-type: none"> • For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 20.
	<p><i>Natural Convection Cooling of a Vacuum Flask</i>: Application Library path Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/vacuum_flask</p>

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.

LOCATION IN USER INTERFACE

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

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.

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 **Surface 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 **Surface emissivity** expression, and used to define the total emissivity:

$$\varepsilon_{\text{tot}}(\theta, \varphi, T, \mathbf{x}) = f_\varepsilon(\theta, \varphi) + \varepsilon(T, \mathbf{x})$$

Select the **Define directional dependence 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.

Surface Emissivity, Directional Emissivity Function, and Diffuse Reflectivity

If the **Wavelength dependence of radiative properties** is **Constant**:

- By default, **Surface 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 surface emissivity on each side** and **Define diffuse reflectivity on each side** check boxes to define specific values for each side. The **Surface emissivity, upside**, **Surface emissivity, downside**, **Diffuse reflectivity, upside**, and **Diffuse reflectivity, downside** should be set.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Surface emissivity** ε (dimensionless) and **Diffuse reflectivity** ρ_d (dimensionless) use values **From material**.
- When **Surface emissivity** is **User defined**, enter a value or expression for **Surface 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 **Surface emissivity** is **User defined for each band**, enter a value for the **Surface 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 surface emissivity 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 diffuse reflectivity 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 **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, $q_{r,\text{net}}$, which is absorbed by the surface.

The net radiative heat flux is defined as the difference between the incoming and outgoing radiative heat fluxes:

$$q_{r,\text{net}} = q_{r,\text{in}} - q_{r,\text{out}}$$

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 \(Radiation in Participating Media Interface\)](#) node instead.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must 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 **Downside** values in the table.

WALL SETTINGS

Select a **Wall type** to define the behavior of the wall: **Gray wall** or **Black wall**. 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 Wall

If **Gray wall** 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

$$I_i = \varepsilon I_b(T) + \frac{1-\varepsilon}{\pi} q_{r, \text{out}}$$

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

$$I_i = \frac{1-\varepsilon}{\pi} q_{r, \text{out}}$$

The default value of the **Surface 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 **Surface 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 surface emissivity on each side** check box to set specific values on each side of the boundary. The **Surface emissivity, upside** and **Surface emissivity, downside** should be set, with the same options as those described above.

Black Wall

If **Black wall** is selected, no user input is required, and the radiative intensity along the discrete incoming directions on this boundary is defined by

$$I_i = I_b(T)$$

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 **PI approximation** is selected as the **Radiation discretization method** for the physics interface, there is additional theory, equations, and variables described in [PI Approximation Theory](#).



- [Participating Medium \(Radiation in Participating Media Interface\)](#)
- [Absorbing-Scattering Medium \(Radiation in Absorbing-Scattering Media Interface\)](#)
- [Theory for Radiation in Participating Media](#)



LOCATION IN USER INTERFACE

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.



LOCATION IN USER INTERFACE

Context Menus

Radiative Beam in Absorbing Media>Opaque Surface

Ribbon

Physics Tab with **Radiative Beam in Absorbing Media** selected:

Boundaries>Opaque Surface

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.



Composite Thermal Barrier, Lumped Thermal System: 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 Solids>Lumped System Connector

Heat Transfer in Fluids>Lumped System Connector

Heat Transfer in Porous Media>Lumped System Connector

Bioheat Transfer>Lumped System Connector

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*>Lumped System Connector

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:

$$-\mathbf{n} \cdot \mathbf{q} = 0, \quad \mathbf{n} \cdot \mathbf{u} \geq 0 \quad (6-19)$$

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:

$$-\mathbf{n} \cdot \mathbf{q} = \rho \Delta H \mathbf{u} \cdot \mathbf{n}, \quad \mathbf{n} \cdot \mathbf{u} < 0 \quad (6-20)$$

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

$$T = T_{\text{ustr}}, \quad \mathbf{n} \cdot \mathbf{u} < 0 \quad (6-21)$$

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 Solids>Flow Conditions>Open Boundary

Heat Transfer in Fluids>Flow Conditions>Open Boundary

Heat Transfer in Porous Media>Flow Conditions>Open Boundary

Bioheat Transfer>Flow Conditions>Open 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*>Open Boundary

Outflow

This node provides a suitable boundary condition for convection-dominated heat transfer at outlet boundaries. In a model with convective heat transfer, this condition states that the only heat transfer occurring across the boundary is by convection. The temperature gradient in the normal direction is zero, and there is no radiation. This is usually a good approximation of the conditions at an outlet boundary in a heat transfer model with fluid flow.

BOUNDARY SELECTION

In most cases, the Outflow node does not require any user input. If required, select the boundaries that are convection-dominated outlet boundaries.



Heat Sink: Application Library path **Heat_Transfer_Module/
Tutorials,_Forced_and_Natural_Convection/heat_sink**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Flow Conditions>Outflow

Heat Transfer in Solids>Flow Conditions>Outflow

Heat Transfer in Fluids>Flow Conditions>Outflow

Heat Transfer in Porous Media>Flow Conditions>Outflow

Bioheat Transfer>Flow Conditions>Outflow

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>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**

LOCATION IN USER INTERFACE

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.

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, $L_{s \rightarrow f}$, and the solid density, ρ_s :

$$v_n = \frac{\mathbf{q}}{\rho_s L_{s \rightarrow f}}$$

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**, $L_{s \rightarrow f}$ (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**.



Upside and downside settings are based on the global normal vector (nx , ny , nz), that always points from downside to upside. A red arrow indicates the direction of the global normal vector on the selected boundaries in the **Graphics** window. 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):

$$v_{\text{mbs}} = \delta_{\text{mbs}} |v_n| h H$$

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 [Stefan Condition](#).



Tin Melting Front: Application Library path **Heat_Transfer_Module/Phase_Change/tin_melting_front**

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**


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.

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** select a **Radiation direction** based on the geometric normal (n_x, n_y, n_z): **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 \(Surface-to-Surface Radiation Interface\)](#) condition.
- Select **Negative normal direction** to specify that the surface radiates in the negative normal direction (**un** vector direction).

- Select **Positive normal direction** if the surface radiates in the positive normal direction (**dn** vector direction).
- Select **Both sides** if the surface radiates on both sides.

When **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands** select a **Radiation direction** for each spectral band: **Opacity controlled** (default), **Negative normal direction**, **Positive normal direction**, or **Both sides**, or **None**. The **Radiation direction** defines the radiation direction for each spectral band similarly as when **Wavelength dependence of radiative properties** is **Constant**.

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 sets the surface radiosity expression corresponding to a blackbody.

- When **Wavelength dependence of radiative properties** is set to **Constant** it defines $J = e_b(T)$ when radiation is defined on one side or $J_u = e_b(T_u)$ and $J_d = e_b(T_d)$ 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 $J_i = FEP_i(T)e_b(T)$ when

radiation is defined on one side or $J_{i,d} = FEP_{i,d}(T_d)e_b(T_d)$ and $J_u = FEP_{i,u}(T_u)e_b(T_u)$ when radiation is defined on both sides.



- When the temperature varies across a pair (for example when a [Thin Layer \(Heat Transfer Interface\)](#) and [Solid \(Heat Transfer in Shells Interface\)](#) condition is active on the same boundary), the temperature used to define the radiosity is evaluated on the side where the surface radiation is defined.
- The blackbody hemispherical total emissive power is defined by $e_b(T) = n^2\sigma T^4$

Graybody Radiation

When **Graybody radiation** is selected it sets the surface radiosity expression corresponding to a graybody.

By default, the **Surface emissivity** ε is defined **From material**. In this case, make sure that a material is defined at the boundary level (materials are defined by default at the domain level). If **User defined** is selected for the **Surface emissivity**, enter another value for ε . 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 **Surface emissivity** ε to set $J = \varepsilon e_b(T)$, or
- When radiation is defined on both sides, define the **Surface emissivity, upside** ε_u and the **Surface emissivity, downside** ε_d on the upside and downside, respectively. The surface radiosity on upside and downside is then defined by $J_u = \varepsilon_u e_b(T_u)$ and $J_d = \varepsilon_d e_b(T_d)$ respectively.

If **Wavelength dependence of 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 **Surface emissivity** ε_i to set $J_i = FEP_i \varepsilon_i e_b(T)$, or
- When radiation is defined on both sides for **Bi** spectral band, define the **Surface emissivity, upside** $\varepsilon_{i,u}$ and the **Surface emissivity, downside** $\varepsilon_{i,d}$ on the upside and

downside, respectively. The surface radiosity on upside and downside is then defined by $J_u = FEP_{Bi}(T_u)\epsilon_{Bi, u}e_b(T_u)$ and $J_d = FEP_{Bi}(T_d)\epsilon_{Bi, d}e_b(T_d)$, respectively.



Set the surface emissivity to a number between 0 and 1, where 0 represents diffuse mirror and 1 is appropriate for a perfect blackbody. The proper value for a physical material lies somewhere in between and can be found from tables or measurements.

User Defined

If **Wavelength dependence of 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](#)

LOCATION IN USER INTERFACE

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.

Radiation Interface) nodes. The numbers on each boundary specify different groups to which the boundary belongs.

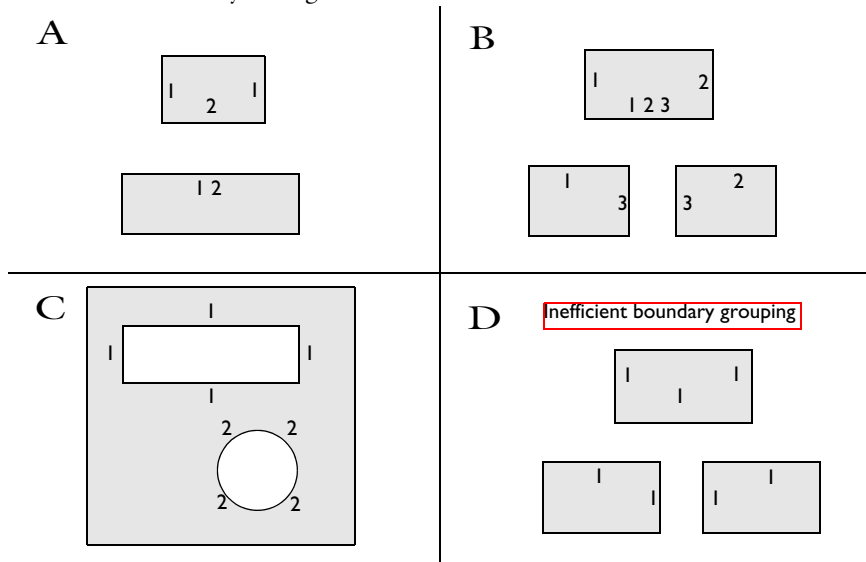


Figure 6-1: Examples of radiation group boundaries.

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 Interface)

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 **PI 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, $q_{r,\text{net,d}}$ and $q_{r,\text{net,u}}$ 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.

WALL SETTINGS

The radiation properties of the boundary and the external radiation intensity should be set in this section.

Surface Emissivity, Transmissivity, and Specular Transmissivity

If the **Wavelength dependence of radiative properties** is **Constant**:

- By default, the **Surface emissivity** ε (dimensionless), **Surface transmissivity** τ_d (dimensionless), and **Surface 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 surface emissivity on each side**, **Define surface transmissivity on each side**, and **Define surface specular transmissivity on each side** check boxes to set specific values for each side. The **Surface emissivity, upside**, **Surface emissivity, downside**, **Surface transmissivity, upside**, **Surface transmissivity, downside**, **Surface specular transmissivity, upside**, and **Surface specular transmissivity, downside** should be set.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, **Surface emissivity** ε (dimensionless), **Surface transmissivity** τ_d (dimensionless), and **Surface specular transmissivity** τ_s (dimensionless) use values **From material**.
- When **Surface emissivity** is **User defined**, enter a value or expression for the **Surface 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 **Surface transmissivity** is **User defined**, enter a value or expression for the **Surface 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 **Surface specular transmissivity** is **User defined**, enter a value or expression for the **Surface specular transmissivity** τ_g . 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 **Surface emissivity** is **User defined for each band**, enter a value for the **Surface 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 surface emissivity on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.
- When **Surface transmissivity** is **User defined for each band**, enter a value for the **Surface 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 surface transmissivity on each side** check box and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.
- When **Surface specular transmissivity** is **User defined for each band**, enter a value for the **Surface 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 surface specular transmissivity 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 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 **Surface specular transmissivity** is only needed when **Discrete ordinates method** is selected.

External Radiation Intensity

The external radiation intensity should be defined when the node 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 **Constant**, the **External radiation intensity** I_{ext} should be specified.

If the **External radiation intensity** is **User defined**, set a value or expression for I_{ext} . Alternatively, choose **Blackbody radiation** to define I_{ext} as $I_{\text{b}}(T)$.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the **External radiation intensity** $I_{\text{ext},k}$ for each spectral band should be specified. Select an option between **Blackbody radiation**, **User defined for each band**, or **User defined distribution**.

If the **External radiation intensity** is **Blackbody radiation**, $I_{\text{ext},k}$ is defined from the blackbody intensity, $I_{\text{b}}(T)$, and the fractional emissive power for each spectral band, FEP_k .

If the **External radiation intensity** is **User defined for each band**, set a value or expression for $I_{\text{ext},k}$ for each spectral band in the table displayed underneath.

If the **External radiation intensity** is a **User defined distribution**, set a value or expression for $I_{\lambda,\text{ext}}$. Any expression is then averaged over each spectral band to obtain a piecewise constant radiation intensity. If the average value of the radiation intensity over each band is known, you can use instead the **User defined for each band** option to avoid 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](#)

LOCATION IN USER INTERFACE

Context Menus

Radiation in Participating Media>Semitransparent Surface

Radiation in Absorbing-Scattering Media>Semitransparent Surface

Radiation in Participating Media>Pairs>Semitransparent Surface

Radiation in Absorbing-Scattering Media>Pairs>Semitransparent Surface

Ribbon

Physics Tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Boundaries>Semitransparent Surface

Pairs>Semitransparent Surface

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.

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 **Surface emissivity** and **Surface 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 $f_\epsilon(\theta) = 0$, $f_\tau(\theta) = 0$, $f_\epsilon(\theta, \varphi) = 0$, or $f_\tau(\theta, \varphi) = 0$.

The other dependencies (temperature T , spatial coordinates \mathbf{x}) are handled separately in the **Surface emissivity** and **Surface transmissivity** expressions, and used to define the total properties:

$$\varepsilon_{\text{tot}}(\theta, \varphi, T, \mathbf{x}) = f_{\varepsilon}(\theta, \varphi) + \varepsilon(T, \mathbf{x})$$

$$\tau_{\text{tot}}(\theta, \varphi, T, \mathbf{x}) = f_{\tau}(\theta, \varphi) + \tau(T, \mathbf{x})$$

Surface Emissivity, Diffuse Reflectivity, and Surface Transmissivity

If **Wavelength dependence of radiative properties** is **Constant**:

- By default, the **Surface emissivity** ε (dimensionless), **Diffuse reflectivity** ρ_d (dimensionless), and **Surface 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 surface emissivity on each side**, **Define diffuse reflectivity on each side**, and **Define surface transmissivity on each side** check boxes to set specific values on each side. The **Surface emissivity, upside**, **Surface emissivity, downside**, **Diffuse reflectivity, upside**, **Diffuse reflectivity, downside**, **Surface transmissivity, upside**, and **Surface transmissivity, downside** should be set.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Surface emissivity** ε (dimensionless), **Diffuse reflectivity** ρ_d (dimensionless), and **Surface transmissivity** τ (dimensionless) use values **From material**.
- When **Surface emissivity** is set to **User defined**, enter a value or expression for the **Surface 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 **Surface transmissivity** is set to **User defined**, enter a value or expression for the **Surface 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 **Surface emissivity** is set to **User defined for each band**, enter a value for the **Surface 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 surface emissivity 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 **Surface transmissivity** is set to **User defined for each band**, enter a value for the **Surface 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 surface transmissivity 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. Select the **Define critical angle on each side** check box to set specific values on each side.

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.

LOCATION IN USER INTERFACE

Context Menus

Surface-to-Surface Radiation>Semitransparent Surface

Ribbon

Physics Tab with **Surface-to-Surface Radiation** selected:

Boundaries>Surface-to-Surface Radiation>Semitransparent Surface

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



$$-\mathbf{n} \cdot \mathbf{q} = \varepsilon \sigma (T_{\text{amb}}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

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.

RADIATION SETTINGS

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 (n_x, n_y, n_z) , that always points from downside to upside. Note that the normal vector $(ht.n_x, ht.n_y, ht.n_z)$ 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

Ribbon

Physics Tab with *interface* as **Heat Transfer in Solids** selected:

Boundaries>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 Solids>Symmetry

Heat Transfer in Fluids>Symmetry

Heat Transfer in Porous Media>Symmetry

Bioheat Transfer>Symmetry

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>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 **PI 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.

	<p>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.</p>
	<p>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.</p>
	<p><i>Steady-State 2D Axisymmetric Heat Transfer with Conduction:</i> Application Library path Heat_Transfer_Module/Tutorials,_Conduction/cylinder_conduction</p>

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids>Temperature

Heat Transfer in Solids>Temperature

Heat Transfer in Fluids>Temperature

Heat Transfer in Porous Media>Temperature

Bioheat Transfer>Temperature

Heat Transfer in Solids>Pairs>Temperature

Heat Transfer in Fluids>Pairs>Temperature

Heat Transfer in Porous Media>Pairs>Temperature

Bioheat Transfer>Pairs>Temperature

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*>**Temperature**
Pairs>*interface*>**Temperature**

Thermal Contact

This node defines correlations for the conductance h at the interface of two bodies in contact. It can be added to pairs by selecting **Pair Thermal Contact** from the **Pairs** menu. Note that in this case the source term is applied on the destination side.

The conductance h is involved in the heat flux across the surfaces in contact according to:

$$\begin{aligned} -\mathbf{n}_d \cdot \mathbf{q}_d &= -h(T_u - T_d) + rQ_b \\ -\mathbf{n}_u \cdot \mathbf{q}_u &= -h(T_d - T_u) + (1 - r)Q_b \end{aligned}$$

where u and d subscripts refer to the upside and downside of the slit, respectively. **Pair Thermal Contact** should be activated on a **Identity Pair** or on a **Contact Pair** where a structural mechanics physics interface defines a contact pair feature.



PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must 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** (). 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.

MATERIAL TYPE

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>Pair Thermal Contact

Heat Transfer in Fluids>Pairs>Pair Thermal Contact

Heat Transfer in Porous Media>Pairs>Pair Thermal Contact

Bioheat Transfer>Pairs>Pair Thermal Contact

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:

Pairs>interface>Pair Thermal Contact

Thermal Insulation

This node is the default boundary condition for all Heat Transfer interfaces. This boundary condition means that there is no heat flux across the boundary:

$$-\mathbf{n} \cdot \mathbf{q} = 0$$

and hence specifies where the domain is well insulated. Intuitively, this equation says that the temperature gradient across the boundary is zero. For this to be true, the temperature on one side of the boundary must equal the temperature on the other side. Because there is no temperature difference across the boundary, heat cannot transfer across it. 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 Solids>Thermal Insulation

Heat Transfer in Fluids>Thermal Insulation

Heat Transfer in Porous Media>Thermal Insulation

Bioheat Transfer>Thermal Insulation

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>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, `s1mat1`), 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 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.


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.

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.

LOCATION IN USER INTERFACE

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, s1mat1), 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 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.

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 `input.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**.



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 (n_x, n_y, n_z) , that always points from downside to upside. Note that the normal vector $(ht.n_x, ht.n_y, ht.n_z)$ 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**

LOCATION IN USER INTERFACE

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)
- 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)



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, s1mat1), 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 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.

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 (n_x, n_y, n_z), that always points from downside to upside. Note that the normal vector ($ht.n_x, ht.n_y, ht.n_z$) may be oriented differently.

See [Tangent and Normal Variables](#) in the *COMSOL Multiphysics Reference Manual*.

BEAM ORIENTATION

Enter a value for the **Beam orientation** \mathbf{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 **Deposited beam power density**, Q_b (SI unit: W/m^2).

For **Built-in beam profiles**, enter a value for the **Deposited beam power** P_0 and the coordinates of the **Beam origin point** \mathbf{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, `s1mat1`), 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.

	<ul style="list-style-type: none">• 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.
	<p>The desired selection for the node may correspond to boundaries with different layered materials.</p> <p>See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single-Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.</p>
	<p>Upside and downside settings can be visualized by plotting the global normal vector (n_x, n_y, n_z), that always points from downside to upside. Note that the normal vector ($ht.n_x, ht.n_y, ht.n_z$) may be oriented differently.</p> <p>See Tangent and Normal Variables in the <i>COMSOL Multiphysics Reference Manual</i>.</p>

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](#)
- [The Heat Transfer Coefficients](#)

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, s1mat1), 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 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.

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 (n_x , n_y , n_z), that always points from downside to upside. Note that the normal vector ($ht.n_x$, $ht.n_y$, $ht.n_z$) 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](#)

LOCATION IN USER INTERFACE

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

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, `s1mat1`), 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.



- 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.

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 (n_x, n_y, n_z), that always points from downside to upside. Note that the normal vector ($ht.n_x, ht.n_y, ht.n_z$) 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.



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, s1mat1), 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

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 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.

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 (n_x , n_y , n_z), that always points from downside to upside. Note that the normal vector ($ht.n_x$, $ht.n_y$, $ht.n_z$) may be oriented differently.

See [Tangent and Normal Variables](#) in the *COMSOL Multiphysics Reference Manual*.

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.

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, s1mat1), otherwise the boundary is marked as not applicable.

INTERFACE SELECTION

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.



- 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.

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 (n_x, n_y, n_z), that always points from downside to upside. Note that the normal vector ($ht.n_x, ht.n_y, ht.n_z$) 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

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
- 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 (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, s1mat1), 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 **Thickness** 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 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.

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** 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**

LOCATION IN USER INTERFACE

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, s1mat1), 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 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.

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

Click the **General inward heat flux** (default), **Convective heat flux**, or **Heat rate** (3D components only) button.

- If **General inward heat flux** is selected, it adds q_0 to the total flux across the selected edges. Enter a value for q_0 to represent a heat flux that enters the layer. For example, any electric heater is well represented by this condition and its geometry can be omitted.
- If **Convective heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{\text{ext}} - T)$. 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**.

- *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, s1mat1), 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](#).
- 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.

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

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 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_1 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_1 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

Click the **General source** (default) or **Heat rate** button.

- If **General source** is selected, enter a value for the distributed heat source, Q_1 in unit power per unit length. A positive Q_1 corresponds to heating while a negative Q_1 corresponds to cooling.
- If **Heat rate** is selected, enter the heat rate P_1 .

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 **Edges** in 3D and **Points** in 2D.

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.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options**.



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.



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, `s1mat1`), 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:

Edges>Surface-to-Ambient Radiation

Symmetry (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, s1mat1), 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:

Edges>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, s1mat1), 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.
	<ul style="list-style-type: none">• 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
	<i>Shell Conduction:</i> 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 \mathbf{q} is defined from the Fourier's law, and this condition sets the temperature gradient to zero across the edge. As the default boundary 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 corresponding to the junction of shells of different layered materials, the continuity of the temperature field across the edge depends on the **Layer type** 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):

- When the **Layer type** is **General**, thermal insulation is implicitly applied, and the temperature field is discontinuous across the edge. 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.
- When the **Layer type** is **Thermally thin approximation**, continuity of the temperature field is implicitly applied.

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, s1mat1), 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 <i>COMSOL Multiphysics Reference Manual</i> .
	Theory for Heat Transfer in Thin Structures
	<i>Shell Conduction</i> : 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:

Edges>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

Select either the **General inward heat flux** (default) or **Convective heat flux** buttons.

- If **General inward heat flux** is selected, it adds q_0 to the total flux across the selected points. Enter a value for q_0 to represent a heat flux that enters the rod.
- If **Convective heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{\text{ext}} - T)$. Enter the **Heat transfer coefficient** h and the **External temperature** T_{ext} . The value depends on the geometry and the ambient flow conditions.



Thin Rod

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids>Thin Rod>Point Heat Flux

More locations are available. For example:

Heat Transfer in Fluids>Thin Rod>Point Heat Flux

Ribbon

Physics Tab with **Thin Rod** selected in the model tree:

Attributes>Point Heat Flux

Point Heat Source

This node, available for 3D components, models a heat source (or sink) that is so small that it can be considered to have no spatial extension.

In theory, the temperature in a point source in 3D is plus infinity (to compensate for the fact that the heat source does not have a spatial extension). The finite element discretization used in COMSOL Multiphysics returns a finite value, but that value must be interpreted in a weak sense.

POINT HEAT SOURCE

Enter the **Point heat source** Q_p in unit power. A positive Q_p corresponds to heating while a negative Q_p corresponds to cooling.

HEAT SOURCE RADIUS

With the Heat Transfer Module, you can model the heat source explicitly and apply it on a ball or disk around the point.

Select the **Specify heat source radius** check box to define the **Heat source radius** R . This setting averages the source on a ball or disk of given radius around the point, and avoids obtaining an increasing temperature shift at the point when meshing finer than this radius. It makes use of the `ballavg` or `diskavg` operator for averaging around the source.



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.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids>Points>Point Heat Source on Axis

Heat Transfer in Fluids>Points>Point Heat Source on Axis

Heat Transfer in Porous Media>Points>Point Heat Source on Axis
Bioheat Transfer>Points>Point Heat Source on Axis

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 on Axis

Surface-to-Ambient Radiation (Thin Rod)

Use this subnode to add surface-to-ambient radiation to points that represent boundaries of a thin rod.

The net inward heat flux from surface-to-ambient radiation is



$$\lim_{\partial S \rightarrow 0} \int_{\partial S} Q ds = d_s \epsilon \sigma (T_{\text{amb}}^4 - T^4)$$

where ϵ is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

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 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
- 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, $T_{ave} = 0.5 \cdot (T_{p1} + T_{p2})$, 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, $T_{ave} = 0.5 \cdot (T_{p1} + T_{p2})$.



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

$$k_R = \frac{16n_r^2\sigma_s T^3}{3\beta_R}$$

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, $T_{ave} = 0.5 \cdot (T_{p1} + T_{p2})$.

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 component.

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**



Composite Thermal Barrier, Lumped Thermal System: 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, $T_{var} = T_{p1}$.

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**, $\phi_{w, \text{ext}}$, 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.
- The detailed definition of the predefined heat transfer coefficients is given in [The Heat Transfer Coefficients](#).
- For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in [Ref. 20](#).

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 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 factor calculation. Each **External Radiation Source** node contributes to the incident radiative heat flux on all spectral

bands, G_{B_i} on all the boundaries where a **Diffuse Surface**, **Diffuse Mirror**, **Opaque Surface** or **Semitransparent Surface** boundary condition is active. The source contribution, $G_{\text{extDir}, i}$, is equal to the product of the view factor of the source by the source radiosity. For radiation sources located on a point, $G_{\text{extDir}, i} = F_{\text{ext}, i} P_{s, i}$. For directional radiative source, $G_{\text{extDir}, i} = F_{\text{ext}, i} q_{0, s}$.

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}_s . The source radiates uniformly in all directions.



\mathbf{x}_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}_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

$q_{s,i} = q_{0,s} FEP_i(T_{\text{sun}})$ where $FEP_i(T_{\text{sun}})$ is the fractional blackbody emissive power over B_i interval at $T_{\text{sun}} = 5780$ 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** $q_{0,s,\lambda}$. The wavelength may be accessed through the `rad.lambda` variable. This distribution is integrated on each spectral band to obtain the source heat flux $q_{0,s,i}$ 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** $q_{0,s}$, to define the source heat flux on the spectral band B_i as $q_{s,i} = FEP_i(T_s)q_{0,s}$ 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 flux** $q_{0,s,i}$ 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**

LOCATION IN USER INTERFACE

Context Menus

Surface-to-Surface Radiation>**External Radiation Source**

Ribbon

Physics Tab with **Surface-to-Surface Radiation** selected:

Global>**External Radiation Source**

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


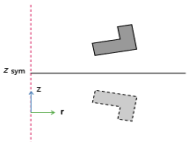


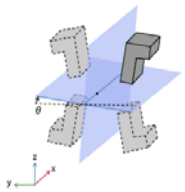
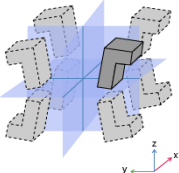

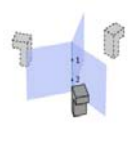
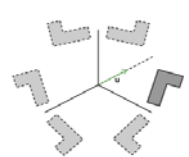
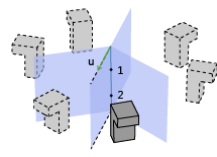
SYMMETRY TYPE	2D	2D AXISYMMETRIC	3D
One Plane			
Two Planes		Not applicable	
Three Planes	Not applicable	Not applicable	

TABLE 6-6: OPTIONS FOR SYMMETRY DEFINITION, EXAMPLE WITH 5 SECTORS

SYMMETRY TYPE	2D	2D AXISYMMETRIC	3D
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.



- [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.

The **Thermal 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 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**



Composite Thermal Barrier, Lumped Thermal System: 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

Ribbon

Physics Tab with **Lumped Thermal System** selected:

Global>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 $\Delta T = T_{\text{condenser}} - T_{\text{evaporator}}$ 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, $T_{\text{ave}} = 0.5 \cdot (T_{p1} + T_{p2})$, 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**, $R_{\text{wall,e}}$, the **Thermal resistance, evaporator wick**, $R_{\text{wick,e}}$, the **Thermal resistance, condenser wick**, $R_{\text{wick,c}}$, and the **Thermal resistance, condenser wall**, $R_{\text{wall,c}}$.

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:

$$R = R_{\text{wall,e}} + R_{\text{wick,e}} + R_{\text{wick,c}} + R_{\text{wall,c}}$$

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_1 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**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$, to be used at initialization, in particular to evaluate the material properties 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

Click to select the **Heat rate** (default), **Convective heat rate**, or **Joule heating** button.

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} :

$$P = \frac{T_{\text{ext}} - T}{R_{\text{conv}}}$$

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:

$$R_{\text{conv}} = \frac{1}{hA}$$

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**, $\phi_{w, \text{ext}}$, 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 [The Heat Transfer Coefficients](#).
- For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in [Ref. 20](#).

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 radiative heat source. The prefix is P.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the radiative heat 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**, $T_{2,\text{init}}$.

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 Heat Source

Ribbon

Physics Tab with **Lumped Thermal System** selected:

Global>Radiative Heat Source

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:

$$P = \epsilon\sigma A(T_{\text{amb}}^4 - T^4)$$

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:

Global>Nodes>Radiative Heat Rate

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**, $T_{1,init}$, and the **Initial temperature at node 2**, $T_{2,init}$, 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 :

$$P = -C \frac{\partial \Delta T}{\partial t} \quad (6-22)$$

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: $\text{J}/(\text{kg}\cdot\text{K})$), 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 6-22](#) 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:

$$P = -C \frac{\partial T}{\partial t} \quad (6-23)$$

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: $\text{J}/(\text{kg}\cdot\text{K})$), 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 6-23](#) 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:

$$P_{p1} = SIT_{p1} + \frac{R_e I^2}{2} - \frac{\Delta T}{R} - C_1 \frac{\partial T_{p1}}{\partial t} \quad (6-24)$$

- p2 side:

$$P_{p2} = -SIT_{p2} + \frac{R_e I^2}{2} + \frac{\Delta T}{R} - C_2 \frac{\partial T_{p2}}{\partial t} \quad (6-25)$$

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 6-24](#) and [Equation 6-25](#) 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:

$$P_{p1} = -\left(Q_{max} - \Delta T \frac{Q_{max}}{\Delta T_{max}}\right) \quad (6-26)$$

- p2 side:

$$P_{p2} = Q_{max} - \Delta T \frac{Q_{max}}{\Delta T_{max}} + R_e I^2 \quad (6-27)$$

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, $T_{ave} = 0.5 * (T_{p1} + T_{p2})$, 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 6-26 and Equation 6-27) 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 6-24 and Equation 6-25 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**, ΔT_{\max} , and the **Electrical resistance**, R_e , used for the expression of the heat rates on each side, following Equation 6-26 and Equation 6-27.

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**, $T_{1,init}$, and the **Initial temperature at node 2**, $T_{2,init}$, 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

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
- Moist Air (Moisture Transport Interface)
- Moisture Source
- 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

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot \mathbf{g}_w = G \quad (7-1)$$

$$\mathbf{g}_w = -(\xi D_w \nabla \phi_w + \delta_p \nabla(\phi_w p_{\text{sat}})) \quad (7-2)$$

$$\xi = \frac{\partial w(\phi_w)}{\partial \phi_w}$$

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: $\text{kg}/(\text{m}^3\cdot\text{s})$) 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.

	<p>The Moisture Transport in Porous Media Interface</p> <p>The Heat and Moisture Transport Interfaces</p> <p>Heat and Moisture</p> <p>Theory for Moisture Transport in Building Materials</p>
	<p>See Building Material Library in the <i>COMSOL Multiphysics Reference Manual</i> for a description of materials containing the thermal and hygroscopic properties required by the Building Material node.</p>
	<p>For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/</p>
	<p><i>Heat and Moisture Transport in a Semi-Infinite Wall</i>: Application Library path Heat_Transfer_Module/Verification_Examples/semi_infinite_wall</p>

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

Ribbon

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. If such user-defined property groups are added, the model inputs appear here.

The default **Temperature** T and **Absolute pressure** p_A are **User defined**. When additional physics interfaces are added to the model, the temperature and absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Heat Transfer in Building Materials** interface is added, you can select **Temperature (ht)** from the list. If a **Laminar Flow** interface is added, you can select **Absolute pressure (spf)** from the list.

If the node was added automatically after selecting the **Heat and Moisture Transport** predefined multiphysics interface, the temperature of the **Heat and Moisture** multiphysics node is used by default and the input field is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** ()

POROUS MATRIX PROPERTIES

The material properties of the porous material should be defined in this section.

The **Porosity**, ε_p (a dimensionless number between 0 and 1), uses by default the value **From material**. For **User defined** the default is 0.

Set the **Permeability**, κ (SI unit: m²), 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.

Finally, 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.

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: m²/s), for the binary diffusion of vapor in air in a free medium. 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 $m_t \cdot T$ and $m_t \cdot pA$.

Then, 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: m²/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.

Finally, set the **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).

LIQUID WATER PROPERTIES

This section contains settings for the definition of convective and capillary transport of liquid water.

First, 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, whereas two 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 to characterize the liquid transport in function of the moisture content.



[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

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 :

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w = G \quad (7-3)$$

$$\mathbf{g}_w = -M_v D \nabla c_v$$

$$c_v = \phi_w c_{\text{sat}}$$

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/m³) is the vapor saturation concentration.
- D (SI unit: m²/s) is the vapor diffusion coefficient in air.
- \mathbf{u} (SI unit: m/s) is the air velocity field.
- G (SI unit: kg/(m³·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 :

$$\rho_g \frac{\partial \omega_v}{\partial t} + \rho_g \mathbf{u} \cdot \nabla \omega_v + \nabla \cdot \mathbf{g}_w = G \quad (7-4)$$

$$\mathbf{g}_w = -\rho_g D \nabla \omega_v$$

$$\omega_v = \frac{M_v c_v}{\rho_g}$$

$$c_v = \phi_w c_{\text{sat}}$$


where ρ_g is the moist air density (SI unit: kg/m³).

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** \mathbf{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 $2.6e-5 \text{ m}^2/\text{s}$ is valid at $T=298 \text{ K}$ and $pA=1 \text{ atm}$. You may set a temperature or pressure-dependent diffusion coefficient through the use of the variables $mt.T$ and $mt.pA$.

	<p>The Moisture Transport in Air Interface</p> <p>The Heat and Moisture Transport Interfaces</p> <p>Heat and Moisture</p> <p>Theory for Moisture Transport in Air</p>
	<p>See Turbulent Mixing to account for eddy diffusivity in moisture convection by turbulent flows.</p>
	<p>For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/</p>
	<p><i>Condensation Detection in an Electronic Device with Transport and Diffusion</i>: Application Library path Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/condensation_electronic_device_transport_diffusion</p>

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

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:

$$D_T = \frac{\nu_T}{Sc_T}$$

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
- 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 relative humidity is continuous across the pair. **Continuity** is only suitable for pairs where the boundaries match.

PAIR SELECTION

Choose the pair on which to apply this condition. A pair has to be created first. See [Identity and Contact Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport 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 relative humidity $\phi_{\text{w,ustr}}$ 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 concentration and upstream concentration. In case of moisture sources or moisture constraints close to the inlet boundary, the inflow condition induces a concentration profile that approaches to the concentration 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:

$$-\mathbf{n} \cdot \mathbf{g}_{\text{w}} = M_{\text{v}}(c_{\text{v}} - c_{\text{ustr}})\mathbf{u} \cdot \mathbf{n}$$

When the **Concentrated species** option is selected in the **Mixture type for moist air** list, this condition may be rewritten as:

$$-\mathbf{n} \cdot \mathbf{g}_{\text{w}} = \rho \left(\omega_{\text{v}} - \frac{M_{\text{v}}c_{\text{ustr}}}{\rho_{\text{g}}} \right) \mathbf{u} \cdot \mathbf{n}$$

to be expressed in function of the mass fraction ω_{v} .

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** $\phi_{\text{w, ustr}}$ from which the upstream vapor concentration can be defined:

$$c_{\text{ustr}} = \phi_{\text{w, ustr}} c_{\text{sat}}(T_{\text{ustr}})$$

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} .



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

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>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:

$$-\mathbf{n} \cdot \mathbf{g}_{\text{w}} = 0$$

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

Context Menus

Moisture Transport in Air>Insulation

Moisture Transport in Building Materials>Insulation

Moisture Transport in Porous Media>Insulation

Ribbon

Physics Tab with *interface* as **Moisture Transport in Air**, **Moisture Transport in Building Materials**, or **Moisture Transport in Porous Media** selected:

Boundaries>Insulation

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: $\text{kg}/(\text{m}^2 \cdot \text{s})$) is defined as:

$$g_{\text{evap}} = \begin{cases} M_v K (c_{\text{sat}} - c_v) & \text{if } c_v > c_{\text{sat}} \text{ or } c_1 > 0 \\ 0 & \text{otherwise} \end{cases}$$

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_1 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:

$$M_v \frac{\partial c_1}{\partial t} = -g_{\text{evap}}$$

with the initial condition:

$$c_l(0) = c_{l, \text{init}}$$

In stationary studies, the initial liquid concentration is used in the definition of the evaporation flux:

$$g_{\text{evap}} = \begin{cases} M_v K (c_{\text{sat}} - c_v) & \text{if } c_v > c_{\text{sat}} \text{ or } c_{l, \text{init}} > 0 \\ 0 & \text{otherwise} \end{cases}$$

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):

$$q_{\text{evap}} = L_v g_{\text{evap}}$$

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 $c_{l, \text{init}}$** .



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, or vapor concentration) 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.

MOISTURE CONTENT


When **Specify** is set to **Relative humidity**, the equation for this condition is

$$c_v = \phi_{w,0} c_{\text{sat}}(T_0)$$

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**).

Alternatively, choose **Vapor concentration** in the **Specify** list and set directly the **Vapor concentration** c_0 .

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.



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>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

Click to select the **General moisture flux** (the default), **Convective moisture flux, concentrations difference**, or **Convective moisture flux, pressures difference** button.

General Moisture Flux

It adds g_0 to the total flux across the selected boundaries. Enter a positive value for g_0 to represent a moisture flux that enters the domain.

Convective Moisture Flux, 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**, $\phi_{w, \text{ext}}$ 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**, $\phi_{w, \text{ext}}$. 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**, $\phi_{w, \text{ext}}$ 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 $g_0 = \beta_p(\phi_{w, \text{ext}} p_{\text{sat}}(T_{\text{ext}}) - \phi_w p_{\text{sat}}(T))$.
- If **Partial vapor pressure** is selected, enter an **External partial vapor pressure**, $p_{v, \text{ext}}$. Convective moisture flux is defined by $g_0 = \beta_p(p_{v, \text{ext}} - \phi_w p_{\text{sat}}(T))$.

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

LOCATION IN USER INTERFACE

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:

$$-\mathbf{n} \cdot \mathbf{g}_w = 0, \quad \mathbf{n} \cdot \mathbf{u} \geq 0 \quad (7-5)$$

For an incoming flow of velocity field \mathbf{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:

$$-\mathbf{n} \cdot \mathbf{g}_w = M_v(c_v - c_{ustr})\mathbf{u} \cdot \mathbf{n}, \quad \mathbf{n} \cdot \mathbf{u} < 0$$

Alternatively the open boundary condition can set a constraint on the vapor concentration for an incoming flow

$$c_v = c_{\text{ustr}}, \quad \mathbf{n} \cdot \mathbf{u} < 0 \quad (7-6)$$

Both conditions are based on the knowledge of the upstream moisture content (from the vapor concentration or the relative humidity and the temperature) at the outer boundary of the virtual domain, but the first one 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** $\phi_{w, \text{ustr}}$ from which the upstream vapor concentration can be defined:

$$c_{\text{ustr}} = \phi_{w, \text{ustr}} c_{\text{sat}}(T_{\text{ustr}})$$

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} .

INFLOW CONDITION

Unfold this section to change the inflow condition:

- The **Flux (Danckwerts)** condition (default) prescribes the flux defined from the upstream vapor concentration 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 vapor concentration and upstream vapor concentration.
- The **Discontinuous Galerkin constraints** option enforces the upstream vapor concentration on the boundary.



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>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:

$$-\mathbf{n} \cdot D\nabla(M_v\phi_w c_{\text{sat}}) = 0$$

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

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

$$\beta = \frac{\delta p_{\text{sat}}}{\mu d_s} = \frac{\delta_p p_{\text{sat}}}{d_s} = \frac{\delta p_{\text{sat}}}{s_d}$$

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](#)

LOCATION IN USER INTERFACE

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: $\text{kg}/(\text{m}^2 \cdot \text{s})$) is defined as:

$$g_{\text{evap}} = M_v K (c_{\text{sat}} - c_v)$$

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):

$$q_{\text{evap}} = L_v g_{\text{evap}}$$

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, \mathbf{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 $-g_{\text{const}}\mathbf{e}_z$ in 2D axisymmetric and 3D models, and $-g_{\text{const}}\mathbf{e}_y$ in 2D models.

Multiphysics Interfaces

The Heat Transfer Module includes predefined multiphysics interfaces for conjugate heat transfer, radiative heat transfer, thermoelectric effect, local thermal nonequilibrium, 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 <http://www.comsol.com/products/specifications/>

The multiphysics interfaces add [Domain Multiphysics Couplings](#) and [Boundary Multiphysics Couplings](#). They predefine the couplings through specific settings in

the multiphysics couplings and in the constituent interfaces to facilitate easy set up of models. These settings are detailed in the following sections:


- [The 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 Local Thermal Nonequilibrium 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](#)

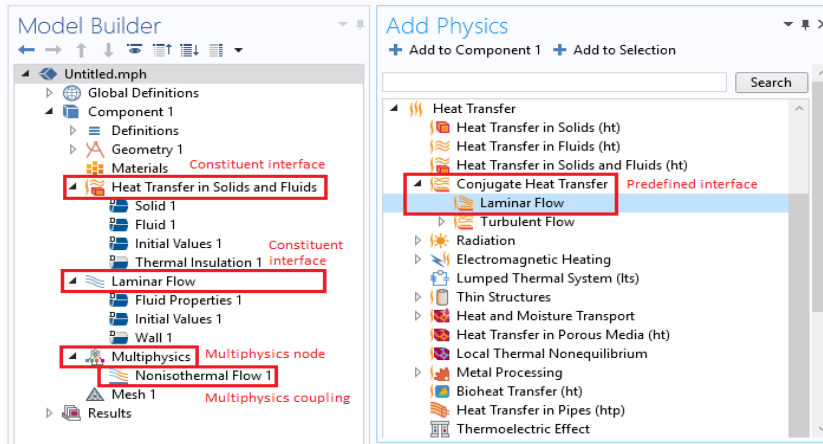
Links to thorough information about these interfaces are given in the corresponding sections.

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.

On 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 \$y\$ Plus Interface](#), [The Turbulent Flow, L-VEL Interface](#), [The Turbulent Flow, \$k\$ - \$\varepsilon\$ Interface](#), and [The Turbulent Flow, Low Re \$k\$ - \$\varepsilon\$ 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, \mathbf{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, \mathbf{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).

AS AN ADD-ON MULTIPHYSICS COUPLING

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 PI

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](#)
- [Settings for Physics Interfaces and Coupling Features](#)
- [Coupling Feature](#)

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	<p>The Domain Selection is the same as that of the participating physics interfaces.</p> <p>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.</p> <p>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.</p>
Electromagnetic Heating	<p>The Domain Selection is the same as that of the participating physics interfaces.</p> <p>The Boundary Selection contains all the boundaries of the participating interfaces.</p> <p>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.</p>

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 Local Thermal Nonequilibrium Interface

In this section:

- [About the Local Thermal Nonequilibrium Interface](#)
- [Coupling Feature](#)
- [Physics Interface Features](#)

About the Local Thermal Nonequilibrium Interface

When the predefined **Local Thermal Nonequilibrium** interface () is added (found under the **Heat Transfer** branch () of the **Model Wizard** or **Add Physics** windows), it combines the Heat Transfer in Solids and the Heat Transfer in Fluids interfaces to model heat transfer in porous media for which the solid and fluid temperatures are not in equilibrium.

Stationary, 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 **Heat Transfer in Fluids** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the multiphysics coupling feature **Local Thermal Nonequilibrium**.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation. A **Solid** model is active by default on all domains.

The Heat Transfer in Fluids interface provides features for modeling heat transfer by conduction, convection, and radiation. A **Fluid** model is active by default on all domains.

AS AN ADD-ON MULTIPHYSICS COUPLING

The Local Thermal Nonequilibrium multiphysics coupling is also available when there is at least one of each of the following interfaces with the default model activated:

- Heat Transfer in Solids, with Solid model
- Heat Transfer in Fluids, with Fluid model

Coupling Feature

See [Local Thermal Nonequilibrium](#) for details about the multiphysics coupling feature.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

The available physics features for [The Heat Transfer in Solids Interface](#) and the [The Heat Transfer in Fluids Interface](#) are listed in [Feature Nodes for the Heat Transfer Interface](#).

Heat Sources

In the heat source features available for the constituent interfaces ([Heat Source](#), [Geothermal Heating](#), and [Bioheat](#)), the user input corresponds to the heat production per total unit volume. It is multiplied by the volume fraction of each phase and added into the corresponding heat equation.

Heat Fluxes

In the heat flux features available for the constituent interfaces ([Heat Flux](#), [Inflow](#)), the user input corresponds to the heat flux per total unit surface. It is multiplied by the volume fraction of each phase and added into the corresponding heat equation. The surface fraction is approximated by the volume fraction.

The Heat and Moisture Transport 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

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 8-7: 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.

ON THE CONSTITUENT PHYSICS INTERFACES

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

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 8-8: 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	<p>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.</p> <p>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, u_l, are automatically set to the variables from the Heat and Moisture multiphysics coupling feature.</p>
Moisture Transport in Porous Media	<p>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.</p>
Heat and Moisture	<p>The Heat Transfer and Moisture Transport interfaces are preselected</p>



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.

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 8-9: 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](#)
- [Physics Interface Features for the Laminar and Turbulent Flow Versions](#)
- [The Moisture Flow, Brinkman Equations Multiphysics Interface](#)
- [Physics Interface Features for the Brinkman Equations Version](#)
- [Preset Studies](#)
- [Coupling Feature](#)

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 , \mathbf{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 γ Plus 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

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.



[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



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	<p>In the Model Input section of the Moist Air default domain feature, the Absolute pressure, p_A, and the Velocity field, \mathbf{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.</p> <p>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.</p> <p>The latent heat sources are automatically handled on boundaries where Wet Surface or Moist Surface features are applied.</p>
Moisture Transport in Air	<p>In the Model Input section of the Moist Air default domain feature, the Absolute pressure, p_A, and the Velocity field, \mathbf{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.</p>
Laminar Flow / Turbulent Flow	<p>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.</p>
Moisture Flow	<p>The Fluid flow and Moisture Transport interfaces are preselected.</p>
Heat and Moisture	<p>The Heat Transfer and Moisture Transport interfaces are preselected.</p>
Nonisothermal Flow	<p>The Fluid flow and Heat Transfer interfaces are preselected.</p>



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 y Plus 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.

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 y PLUS

The available physics features for [The Turbulent Flow, Algebraic \$y\$ Plus 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\$ - \$\epsilon\$ 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\$ - \$\omega\$ 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\$ - \$\epsilon\$ 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.

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 8-13: 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	<p>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.</p> <p>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.</p> <p>The latent heat sources are automatically handled on boundaries where Wet Surface or Moist Surface features are applied.</p>
Moisture Transport in Porous Media	<p>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.</p>

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](#)
- [Coupling Feature](#)

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.

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
- Local Thermal Nonequilibrium
- 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**

LOCATION IN USER INTERFACE

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** subfeature 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:

$$q_{\text{evap}} = L_v g_{\text{evap}}$$

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


LOCATION IN USER INTERFACE

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 ht_rpm1.

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).


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 `htasm1`.

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).


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 i^{th} beam intensity (SI unit: W/m^2), solution of the Beer-Lambert equation.

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 `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.

Local Thermal Nonequilibrium

Use the **Local Thermal Nonequilibrium** multiphysics coupling () to account for heat transfer in porous domains where the solid and fluid temperatures are not in equilibrium. This is achieved by coupling the heat equations in the solid and fluid subdomains through a transfer term proportional to the temperature difference between the fluid and the solid. The corresponding heat equations in the solid and in the fluid subdomains read

$$\theta_s \rho_s C_{p,s} \frac{\partial T_s}{\partial t} + \theta_s \rho_s C_{p,s} \mathbf{u}_s \cdot \nabla T_s = \nabla \cdot (\theta_s k_s \nabla T_s) + q_{sf}(T_f - T_s)$$

$$(1 - \theta_s) \rho_f C_{p,f} \frac{\partial T_f}{\partial t} + (1 - \theta_s) \rho_f C_{p,f} \mathbf{u}_f \cdot \nabla T_f = \nabla \cdot ((1 - \theta_s) k_f \nabla T_f) + q_{sf}(T_s - T_f)$$

with the following material properties:

- θ_s is the solid volume fraction.
- ρ_s and ρ_f are the solid and fluid densities.
- $C_{p,s}$ and $C_{p,f}$ are the solid and fluid heat capacities at constant pressure.
- k_s and k_f are the solid and fluid thermal conductivities.
- q_{sf} is the interstitial convective heat transfer coefficient.
- \mathbf{u}_s and \mathbf{u}_f are the solid and fluid velocity vectors.

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

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



See also [Local Thermal Nonequilibrium](#) in [Theory for Heat Transfer in Porous Media](#).

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `1tne1`.

COUPLED INTERFACES

This section defines the physics involved in the **Local Thermal Nonequilibrium** multiphysics coupling.

Select the **Heat transfer in solids** interface associated to the solid temperature dependent variable. Select the **Heat transfer in fluids** interface associated to the fluid temperature-dependent variable.

LOCAL THERMAL NONEQUILIBRIUM SETTINGS

Enter a **Solid volume fraction** θ_s (dimensionless). The default value is 0.5.

Select an **Interstitial convective heat transfer coefficient: Spherical pellet bed, General configuration**, or **User defined** (the default).

Spherical Pellet Bed

In this particular configuration, the interstitial convective heat transfer coefficient can be directly expressed as a function of the average pellet radius r_p and the fluid-to-solid Nusselt number for which the fluid dynamic viscosity μ is needed.

Enter a value for the **Average pellet radius** r_p (SI unit: m). Default value is $5e-4$ m.

The default **Dynamic viscosity** μ (SI unit: Pa-s) is used **From material**. In the list, choose **User defined** to enter another value or expression. When the dynamic viscosity is set in the Heat Transfer in Fluids interface, it also appears in the list.



The Heat Transfer in Fluids interface defines the dynamic viscosity if either **Moist air** is selected as **Fluid type** in the **Thermodynamics, Fluid** section, or the **Convectively Enhanced Conductivity** subnode is added under **Fluid** node.

General Configuration

The interstitial convective heat transfer coefficient is expressed as the product of the specific surface area a_{sf} and the interstitial heat transfer coefficient h_{sf} .

Enter a value for the **Specific surface area** a_{sf} (SI unit: 1/m).

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)).


LOCATION IN USER INTERFACE

Context Menus

Multiphysics>Local Thermal Nonequilibrium

when the **Heat Transfer in Solids** interface with **Solid** feature is added together with the **Heat Transfer in Fluids** interface with **Heat Transfer in Fluids** feature.

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 **Screen** and **Interior Fan** feature 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 $(1 - m_t.s1)$, where the $m_t.s1$ 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 TURBULENCE MODEL

This section is available for free media flow, when the 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


LOCATION IN USER INTERFACE

Context Menus

Multiphysics>Moisture Flow

- when any version of a **Single-Phase Flow** (or **Brinkman Equations, Free and Porous Media Flow**) interface with **Fluid Properties** feature is active together with a **Moisture Transport** interface with **Moist Air** feature.
- when any version of a **Single-Phase Flow** (or **Brinkman Equations, Free and Porous Media Flow, Darcy's Law**) interface with **Fluid and Matrix Properties** feature is active together with a **Moisture Transport** interface with **Hygroscopic Porous Medium** feature.

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, i.e. 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** Re_{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 be 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: $1/\text{K}$), 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 $\mathbf{J}_e = -\sigma \nabla T$ 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.

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.



- 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.

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.


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.

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 overridden on each domain with the [Opacity \(Surface-to-Surface Radiation Interface\)](#) boundary node.



The opacity setting is used when the **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 **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 **Radiation direction** is **Both sides**, and this setting is always ignored.


LOCATION IN USER INTERFACE

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 an interface (generally liquid-air) depends on the concentration of a species or on the temperature distribution. In the case of temperature dependence, the Marangoni effect is also called thermo-capillary convection. It is of primary importance in the fields of welding, crystal growth, and electron beam melting of metals.

The Marangoni effect is a shear stress which depends on the tangential temperature gradient and should be implemented as such. It has the following contribution described by forces induced on the fluid/fluid interface:

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

where γ is the temperature derivative of the surface tension σ (N/(m·K)). Note that this formulation is intended for laminar flow regimes only.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `me1`.

SURFACE TENSION

Select a **Surface tension coefficient type**: **Library coefficient**, **liquid/gas interface**, or **User defined** (the default).

- For **Library coefficient**, **liquid/gas interface** choose an option from the **Library surface tension coefficient** list.
- For **User defined** enter a **Surface tension coefficient** σ (SI unit: N/m).

MARANGONI EFFECT

This section defines the physics involved in the multiphysics coupling. The **Fluid flow** and **Heat transfer** lists include all applicable physics interfaces.

The default values depend on how this coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Heat Transfer in Fluids** is deleted — then the **Heat transfer** list defaults to **None** as there is nothing to couple to.



If a physics interface is deleted and then added to the model again, then in order to 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*.



Marangoni Effect: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/marangoni_effect**

LOCATION IN USER INTERFACE

Context Menus


Multiphysics>Marangoni Effect

when any of the following interface is added together with **Heat Transfer in Solids** (or another version of the Heat Transfer Interface):

Single-Phase Flow (any version)

Porous Media and Subsurface Flow, Brinkman Equations

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.



- 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.

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.

I n d e x

- A** absolute pressure 51, 196, 437, 442, 462, 470, 485, 496, 531, 537, 602, 663, 679, 714
- absorbing medium 91
- absorbing medium (node) 419
- absorbing-scattering medium 91
- absorbing-scattering medium (node) 420
- acceleration of gravity 48, 313
- activation energy 47, 183, 456, 503
- apparent heat capacity 194
- Application Libraries window 38
- application library examples
 - ambient settings 136
 - bioheat 426
 - bioheat transfer interface 378
 - biological tissue 428, 504
 - boundary heat source 515
 - boundary thermoelectric effect 800
 - building material 698
 - conductive thermal resistor 662
 - consistent stabilization 95
 - continuity 516
 - continuity on interior boundary 517
 - convectively enhanced conductivity 435
 - diffuse surface 527
 - electromagnetic heating 779
 - external radiation source 669
 - external terminal 674
 - fluid 440
 - heat and moisture 782
 - heat flux 537
 - heat flux (heat transfer in thin shells interface) 631
 - heat source 449
 - heat transfer in porous media 374
 - heat transfer in thin shells 381
 - highly conductive layers 217
 - inflow (moisture transport) 709
 - initial values (heat transfer in thin shells interface) 550
 - insulation/continuity (heat transfer in thin shells interface) 649
 - isothermal domain 460
 - isothermal domain interface 555
 - lumped system connector 565
 - marangoni effect 807
 - moist air 464
 - moist air (moisture transport) 704
 - moist surface 711
 - moisture content 713
 - moisture flow 791
 - moisture flux 715
 - nonisothermal flow 798
 - opaque 469
 - opaque surface 564
 - open boundary 567
 - open boundary (moisture transport) 716
 - optically thick participating medium 471
 - outflow 568
 - out-of-plane heat flux 472
 - out-of-plane radiation 474
 - periodic condition 568, 572
 - phase change material 484
 - point heat source 654
 - radiation in participating media 402
 - radiation in participating media (rpm interface) 480
 - solid 500
 - symmetry 591

- temperature (heat transfer in thin shells interface) 647
- temperature (heat transfer interface) 593
- thermal contact 307, 598
- thermal expansion 798
- thin layer 609
- thin moisture barrier 719
- translational motion 509
- wet surface 720
- arterial blood temperature 54, 425
- average gas particle diameter 47
- average particle diameter 47, 186
- axisymmetric geometries 256, 397, 578
- azimuth 59, 256
- azimuthal sectors 256
- B**
 - beam orientation 47, 518, 545, 613
 - bioheat (node) 91, 425
 - bioheat transfer interface 349, 378
 - theory 181
 - biological tissue (node) 91, 378, 426
 - black walls 563
 - blackbody radiation 544, 574, 584
 - blackbody radiation intensity 49, 259, 475, 783
 - boundary conditions
 - heat equation, and 85
 - heat transfer coefficients, and 311
 - boundary flux variables (heat transfer interface) 72
 - boundary heat source (node) 513
 - boundary heat source variable 76
 - boundary interface nodes 611
 - boundary multiphysics nodes 723, 802
 - boundary nodes 511, 707
 - Boussinesq approximation 286
 - Brinell hardness 49, 305
 - building material (moisture transport interface) 416
 - building material (node) 91
 - heat transfer interface 376–377, 429, 783
 - moisture transport interface 696, 783
 - bulk velocity 313
- C**
 - change thickness (node) 91
 - characteristic length 313
 - Charron's relation 306, 597
 - coefficient of thermal expansion 56, 161, 314, 491
 - common settings 31
 - concentration 46, 175, 443, 463
 - conduction, definition 164
 - conductive heat flux variable 69
 - conductive heat flux vector 85
 - conductive thermal resistor (node) 658
 - conjugate heat transfer (multiphysics interface) 728
 - conjugate heat transfer (settings) 373
 - consistent stabilization (settings) 358, 367, 369
 - consistent stabilization method 95
 - constraints, Galerkin 593
 - continuity (moisture transport interface) 707
 - continuity (node) 516–517
 - heat transfer in shells 640
 - continuity on interior boundary (node) 403, 407, 517
 - convection, definition 164
 - convection, natural and forced 312
 - convective heat flux variable 73
 - convective heat flux variable, cflux 70
 - convective thermal resistor (node) 662
 - convectively enhanced conductivity (node) 434
 - Cooper-Mikic-Yovanovich (CMY) corre-

- lution 304, 596
 - cross sectional area 46, 358, 368
 - cross sectional perimeter 52, 358, 368
 - cross-section (node) 91, 435
 - crosswind diffusion
 - definition 95
 - heat transfer, and 95
 - crosswind diffusion, consistent stabilization method 95
- D**
- Dalton's law 175
 - damage integral analysis discretization (settings) 359
 - Darcy number 47, 186
 - density, blood 58, 426
 - dependent variables (settings) 360, 367
 - deposited beam power (node) 93, 518
 - diffuse gray radiation model 396
 - diffuse mirror (node) 519
 - diffuse spectral radiation model 396
 - diffuse surface (node) 521
 - diffuse-gray surfaces 244, 247
 - diffuse-spectral surface 250
 - Dirac pulse 196
 - direct area integration, axisymmetric geometry and 256
 - discontinuous Galerkin constraints 593
 - discrete ordinates method (DOM) 264, 402, 407, 422, 476
 - discretization (settings) 359, 367, 369, 378, 380–381, 384, 389, 412
 - dispersion tensor 47, 505
 - dispersivities, porous media 57, 505
 - documentation 36
 - domain flux variables 69
 - domain heat source variables 75
 - domain multiphysics nodes 723, 779
 - domain nodes 418, 696
 - dry solid density 198, 432
 - dry solid specific heat capacity 198, 432
 - dry solid thermal conductivity 198, 431
- E**
- edge nodes 629
 - effective thermal conductivity 50, 187, 191, 198, 429, 466, 488–489, 505
 - effective volumetric heat capacity 56, 187, 198, 500
 - electric currents interface 747
 - electromagnetic heat source 52
 - electromagnetic heat source (multiphysics coupling) 773
 - electromagnetic heating (multiphysics coupling) 746, 772, 774–775, 779
 - Electromagnetic heating, layered shell (multiphysics coupling) 802
 - elevation 256
 - emailing COMSOL 39
 - emission, radiation and 259
 - energy rates 335
 - evaluating view factors 253
 - external radiation source (node) 664
 - External temperature 93
 - external temperature (node) 527
 - external terminal (node) 673
- F**
- Favre average 285
 - film thickness 219
 - fluid
 - htlsh interface 599
 - node (porous medium) 441
 - fluid (node) 91, 436
 - Fourier's law 438, 443, 486, 499
 - fracture (node) 528
 - frames, conversions between 337
 - frames, moving 89
 - frequency factor 46, 183, 456, 503
- G**
- Galerkin constraints, heat transfer 593
 - gap conductance 49, 305, 594

- geothermal heating (node) 91, 445
- global nodes 658
- Grashof number 48, 313
- gravity 48, 313
- gray walls 562
- graybody radiation 249, 544, 574
- gray-diffuse parallel plate model 306
- guidelines, solving surface-to-surface radiation problems 112

H harmonic perturbation (node)

- heat transfer interface 533

heat and moisture (multiphysics coupling) 756, 771, 780

heat and moisture flow (multiphysics interface) 763, 768

heat and moisture transport (multiphysics interface) 375–377, 751–752, 754

heat balance 165

heat capacity at constant pressure 46, 161

heat flux (node)

- fracture subnode 632
- heat transfer in shells interface 629
- heat transfer interface 534
- thin film subnode 632
- thin layer subnode 632

heat flux, interface (node)

- heat transfer in shells 614

heat flux, theory 332

heat pipe (node) 674

heat rate 447, 514, 535, 537, 598

heat rate (node) 678

heat rate source (node) 680

heat source (node) 91, 540

- heat transfer 446
- heat transfer in shells interface 538
- heat transfer in shells, edge condition 635

heat source, interface (node)

- heat transfer in shells 611, 617

heat sources

- defining as heat rate 447
- line and point 638
- local thermal non-equilibrium 749
- thin layer 540

heat transfer coefficients

- theory 312

heat transfer in building materials interface 349

heat transfer in films interface 350, 383

heat transfer in fluids (node) 372–373

heat transfer in fluids interface 349, 372, 737, 741, 743, 756

- theory 171

heat transfer in fractures interface 350, 387

heat transfer in moist porous media interface 349, 375

- theory 191

heat transfer in porous media interface 349, 373–374

- theory 186

heat transfer in shells interface 350, 379

heat transfer in shells interfaces 350

heat transfer in solids (node) 373

heat transfer in solids and fluids interface 373

heat transfer in solids interface 349, 371, 735, 737, 740, 743, 747, 749, 756

- theory 170

heat transfer interface 349, 376–377

heat transfer interfaces 349

Heat Transfer Module 66

heat transfer with radiation in absorbing media (multiphysics interface) 742

heat transfer with radiation in absorb-

- ing-scattering media (multiphysics coupling) 784
 - heat transfer with radiation in absorbing-scattering media (node) 784
 - heat transfer with radiation in participating media (multiphysics coupling) 783
 - heat transfer with radiation in participating media (node) 783
 - heat transfer with radiation in participating medium (multiphysics coupling) 738, 741
 - heat transfer with radiation in participating medium (multiphysics interface) 736, 739
 - heat transfer with radiative beam in absorbing media (multiphysics coupling) 743, 786
 - heat transfer with radiative beam in absorbing media (node) 786
 - heat transfer with surface-to-surface radiation (multiphysics coupling) 735, 803
 - heat transfer with surface-to-surface radiation (multiphysics interface) 733
 - heat transfer with surface-to-surface radiation (node) 803
 - heat transfer with surface-to-surface radiation interface 349
 - heat transfer, and streamline diffusion 95
 - Heaviside function 196
 - hemicubes, axisymmetric geometry and 256
 - hygroscopic porous medium (moisture transport interface) 415
 - hygroscopic porous medium (node) 783, 792
- I** immobile fluids (node) 91, 449
- incident intensity (node) 403, 407–408, 543
 - inconsistent stabilization (settings) 359, 367, 369
 - inconsistent stabilization methods 96
 - induction heating (multiphysics interface) 774
 - infinite elements (node) 91
 - inflow (node) 547, 708
 - initial values
 - node (htlsh interface) 548
 - initial values (node) 371–372, 374–378, 380, 384, 388
 - heat transfer in shells interface 548
 - heat transfer interface 452
 - moisture transport interface 702
 - radiation in participating media interface 453
 - radiative beam in absorbing media interface 453
 - surface-to-surface radiation interface 550
 - insulation (node) 709
 - insulation/continuity (node) 380, 384, 388
 - internal boundary heat flux variables 73
 - internal energy 47, 332
 - internet resources 36
 - inward heat flux 85
 - irreversible transformation (node) 91, 456
 - isothermal domain (node) 361, 459
 - isothermal domain (settings) 358, 361–362
 - isothermal domain interface (node) 362, 552
 - isotropic diffusion 96, 369
- J** Joule heating (multiphysics interface) 772
- K** Karman constant 290

- Kays-Crawford models 289
- knowledge base, COMSOL 39
- L**
 - laminar flow interface 727, 757, 763
 - laser heating (multiphysics interface) 773
 - latent heat of evaporation 51, 199, 429
 - latitude 256, 666
 - layer opacity (node) 556
 - layer thickness 47, 211–212, 216
 - Layered thermal expansion 805
 - Legendre coefficient 46, 269, 273, 424, 479
 - line and point heat source variables 76
 - line heat source (node) 557
 - heat transfer interface 638
 - line heat source on axis (node) 557
 - heat transfer 557
 - line heat source variable 76
 - local thermal non-equilibrium (multiphysics coupling) 749
 - local thermal nonequilibrium (multiphysics coupling) 787
 - local thermal non-equilibrium (multiphysics interface) 374, 748
 - local thermal non-equilibrium interface theory 188
 - longitude 256, 666
 - lumped system connector (node) 565
 - lumped thermal system interface 392
- M**
 - Marangoni effect (multiphysics coupling) 806
 - mean effective thermal conductivity 499
 - mean effective thermal diffusivity 500
 - mechanisms of heat transfer 164
 - metabolic heat source 53, 426
 - microwave heating (multiphysics interface) 775
 - Mikic elastic correlation 305, 596
 - moist air 444, 603
 - moist air (moisture transport interface) 414
 - moist air (node) 91, 462, 783
 - moist porous medium (node) 465, 783
 - moist surface (node) 710
 - moisture content 55, 174, 444, 463, 603
 - moisture content (node) 712
 - moisture diffusivity 47, 280, 282, 697
 - moisture flow (multiphysics coupling) 762, 771, 789
 - moisture flow (multiphysics interface) 757, 761
 - moisture flow interface
 - theory 291
 - moisture flux (node) 713
 - moisture source (node)
 - moisture transport interface 705
 - moisture storage capacity 59, 282, 696
 - moisture storage function 55
 - moisture transfer coefficient 56
 - moisture transport in air interface 355, 414, 760, 762, 767
 - moisture transport in building materials interface 356, 415–416
 - moisture transport in porous media interface 356
 - moisture transport interface 783
 - moisture transport interfaces 356
 - moving frames 89
 - moving mesh, heat transfer, and 197, 483
 - MPH-files 38
 - multiphysics couplings
 - electromagnetic heating (node) 779
 - Electromagnetic heating, layered shell (node) 802
 - heat and moisture (node) 780
 - local thermal nonequilibrium (node) 787

- Marangoni effect (node) 806
 - moisture flow (node) 789
 - nonisothermal flow (node) 792
 - thermal expansion (node) 798
 - thermoelectric effect (node) 798
 - mutual irradiation 48, 248
- N**
- natural and forced convection 312
 - nodes, common settings 31
 - nonisothermal flow (multiphysics coupling) 98, 730, 771, 792
 - nonisothermal flow (multiphysics interface) 727
 - nonisothermal flow interface
 - theory 284
 - normal conductive heat flux variable 72
 - normal convective heat flux variable 73
 - normal total energy flux variable 73
 - normal total enthalpy flux variable 73
 - Nusselt number 51, 190, 312, 434, 788
- O**
- opaque (node) 468
 - opaque surface (node) 402, 407, 557, 561
 - open boundary (moisture transport interface) 715
 - open boundary (node) 566
 - optically thick participating medium (node) 469
 - outflow (moisture transport interface) 717
 - outflow (node) 567
 - out-of-plane flux variables 71
 - out-of-plane heat flux (node) 471
 - out-of-plane heat transfer
 - theory 308
 - thin shells theory 215
 - out-of-plane inward heat flux variable 71
 - out-of-plane radiation (node) 473
 - out-of-plane thickness 47, 211–212, 216, 219, 366, 616
- P**
- PI approximation 268, 402, 407, 422, 476
 - pair boundary heat source (node) 513
 - pair thermal contact (node) 594
 - participating media, radiative heat transfer 258
 - Peltier effect 296, 798, 808
 - Pennes' approximation 181, 426
 - performance index 52, 403, 408
 - perfusion rate, blood 59, 425
 - periodic condition (node) 568
 - rpm interface 569
 - phase change interface (node) 570
 - phase change material (node) 91, 480
 - phase transitions 481
 - physical model (settings) 357, 366, 368, 372, 374–375
 - physics interfaces, common settings 31
 - point heat flux (node)
 - thin rod subnode 652
 - point heat source (node)
 - heat transfer 653
 - point heat source on axis (node) 654
 - point heat source variable 76
 - point nodes 652
 - points
 - heat flux 652
 - surface-to-ambient radiation 655
 - temperature 656
 - porous matrix
 - node (porous medium) 484
 - porous medium
 - htlsh interface 528
 - porous medium (moisture transport interface) 696
 - porous medium (node) 91, 374, 387
 - Prandtl number 52, 190, 289, 313
 - prescribed radiosity (node) 573
 - pressure work (node) 491

- R** radiation
 - axisymmetric geometries, and 256, 397, 578
 - participating media 258
 - radiation branch (settings) 395
 - radiation group (node) 577
 - radiation in participating media (node)
 - rpm interface 475
 - radiation in participating media interface
 - 401
 - theory 258
 - radiation intensity, blackbody 49, 259, 475, 783
 - radiation, definition 165
 - radiative conductance 49, 306, 596, 627
 - radiative heat flux variable 73
 - radiative heat source (node) 681
 - radiative heat, theory 237
 - Radiative Intensity Source 91
 - radiative out-of-plane heat flux variable
 - 71
 - radiative source (node) 492
 - radiative thermal resistor (node) 683
 - radiative transfer equation 259
 - radiogenic heating per mass 53, 445
 - radiosity 50, 237, 239, 248, 252, 520–521, 557, 573, 585, 591, 665
 - radiosity expressions 574
 - radiosity method 240, 247
 - ratio of specific heats 56, 438, 443–444, 482–483
 - Rayleigh number 53, 313
 - reflectivity 58, 237–238, 262
 - refractive index 51, 241, 243, 260, 262, 275, 403, 408, 470
 - relative humidity 58, 174, 199, 276, 279, 282, 429, 444, 463, 603, 696, 703
 - Reynolds number 53, 312, 794
 - extended Kays-Crawford 289
 - Rodriguez formula 261
- S** scattering, radiation and 259
 - sectors, azimuthal 256
 - Seebeck coefficient 54, 296, 800, 809
 - Seebeck effect 296
 - Semi-transparent surface 93
 - semi-transparent surface (node) 585
 - sensible enthalpy 163
 - Settings for the Heat Transfer in Shells
 - Interface 363
 - shape memory alloy (node) 91, 494
 - shell thickness 211–212, 216, 219
 - solar position 256, 665
 - solid (node) 91, 371, 379, 498
 - htlsh interface 604
 - solver settings 105
 - solving surface-to-surface radiation
 - problems 112
 - source terms, bioheat 425
 - specific heat capacity
 - biological tissue 428
 - blood 46
 - damaged tissue 46, 458, 504
 - definition 161
 - fluid 46, 436
 - fracture 533
 - phase change 482–483
 - porous media 191, 488
 - solids 495, 498
 - thin rod 651
 - specific heat, blood 425
 - specific surface area 46, 190, 788
 - spectral band 46, 242, 468, 520, 556–557, 573, 585, 664
 - stabilization techniques
 - crosswind diffusion 95, 358, 367, 369, 412

- isotropic diffusion 359, 367, 369, 412
- streamline diffusion 358, 367, 369, 412
- standard settings 31
- stationary study 349
- Subsurface Flow Module 446, 452
- subsystem definition (node) 684
- subsystem instance (node) 685
- sun position 669
- surface emissivity 56, 248, 473, 519, 543–544, 562, 575, 589, 597, 620, 627, 642, 655
- surface reflectivity 58
- Surface-to-ambient radiation 93
- surface-to-ambient radiation (node) 589
 - fracture subnode 641
 - thin film subnode 641
 - thin layer subnode 641
 - thin rod subnode 655
- surface-to-ambient radiation, interface (node) 620
- surface-to-surface radiation interface 395
 - theory 237
- symmetry (moisture transport interface) 718
- symmetry (node)
 - ht interface 591
 - rpm interface 591
- T** technical support, COMSOL 39
- temperature (node)
 - heat transfer in shells interface 645
 - heat transfer interface 592
 - thin rod subnode 656
- temperature, interface (node)
 - heat transfer in shells 623, 625
- temperature, LTS interface (node) 686
- theory
 - bioheat transfer interface 181
 - conjugate heat transfer multiphysics interface 284
 - heat flux and balance 332
 - heat transfer coefficients 311–312
 - heat transfer in fluids interface 171
 - heat transfer in moist porous media interface 191
 - heat transfer in porous media interface 186
 - heat transfer in solids interface 170
 - isothermal domain feature 202
 - local thermal non-equilibrium interface 188
 - material and spatial frames 336
 - moist air fluid type 174
 - moisture flow interface 291
 - nonisothermal flow 284
 - out-of-plane heat transfer 308
 - radiation in participating media interface 258
 - radiative heat transfer in transparent media 237
 - surface tension coefficients 331
 - surface-to-surface radiation interface 237
 - thermal contact 303
 - thermoelectric effect interface 295
 - thin rod feature 221
 - thermal capacitor (node) 686
 - thermal conductivity 50, 164
 - thermal conductivity components, thin shells 209
 - thermal conductivity supplement 46, 198, 431
 - thermal conductivity, frames and 339
 - thermal conductivity, mean effective 499
 - thermal contact (node) 594
 - theory 303

thermal damage (node) 501
 thermal diffusivity 56, 314, 500
 thermal dispersion (node) 505
 thermal expansion (multiphysics coupling) 798
 thermal expansion coefficient 56, 161
 thermal friction 306
 thermal insulation 85
 thermal insulation (node) 371–372, 374–378, 599, 648
 thermal mass (node) 688
 thermoelastic damping (node) 506
 thermoelectric effect (multiphysics coupling) 746, 798
 thermoelectric effect (multiphysics interface) 744
 thermoelectric effect interface theory 295
 Thermoelectric effect, layered shell 808
 thermoelectric module (node) 689
 thickness 357, 368
 out-of-plane 47, 508, 616, 619, 622, 631, 637
 shell 47
 thin film 47
 thin layer 47
 thickness (node)
 heat transfer interface 507
 thin conductive layers, definition 216
 thin film (node) 599
 thin layer (node) 604
 thin layer, general 207
 thin moisture barrier (node) 718
 thin rod (node) 650
 Thomson effect 296
 time zone 256, 666
 time-dependent study 349
 total boundaries area 46
 total energy flux variable 71
 total enthalpy flux variable, thflux 71
 total heat flux variable 69
 total internal energy 47
 total normal heat flux variable 72
 total out-of-plane heat flux variable 72
 translational motion (node) 508
 transparent media refractive index 51
 turbulence modeling constant 46, 294
 turbulent conjugate heat transfer theory 285
 turbulent flow, algebraic yPlus interface 728, 757, 763
 turbulent flow, k-e interface 728, 757, 764
 turbulent flow, k-omega interface 728, 758, 764
 turbulent flow, low Re k-e interface 728, 758, 764
 turbulent flow, L-VEL interface 728, 757, 764
 turbulent flow, Spalart-Allmaras interface 728, 758, 764
 turbulent flow, SST interface 728, 758, 764
 turbulent heat flux variable 70
 turbulent mixing (node)
 moisture transport interface 706
 turbulent nonisothermal flow interfaces theory 285
 turbulent Prandtl number 52, 289, 794
 turbulent Schmidt number 791

V

Vapor diffusion coefficient in air 47
 vapor diffusion coefficient in air 276, 279, 703
 vapor mass fraction 56, 444, 463, 603
 vapor permeability 56, 199, 282, 429, 696
 vapor permeability of still air 56
 vapor resistance factor 57

vapor saturation concentration 47, 276,
279, 703
vapor saturation pressure 199, 282, 429,
696
Vickers correlation coefficient 304, 596
Vickers size index 304, 596
view factors 48, 253, 521, 664
viscous dissipation (node) 509
volume reference temperature 94
volumetric heat capacity 56, 500

W water content 198, 432
websites, COMSOL 39
wet surface (node) 719

Y Young's modulus 47

Z zenith 57, 256

