

Heat Transfer Module

User's Guide

Heat Transfer Module User's Guide

© 1998–2024 COMSOL

Protected by patents listed on www.comsol.com/patents, or see Help > About COMSOL Multiphysics on the File menu in the COMSOL Desktop for less detailed lists of U.S. Patents that may apply. Patents pending.

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

COMSOL, 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 6.3

Contact Information

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

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

- Support Center: www.comsol.com/support
- Product Download: www.comsol.com/product-download
- Product Updates: www.comsol.com/product-update
- COMSOL Blog: www.comsol.com/blogs
- Discussion Forum: www.comsol.com/forum
- Events: www.comsol.com/events
- COMSOL Video Gallery: www.comsol.com/videos
- Support Knowledge Base: www.comsol.com/support/knowledgebase
- Learning Center: <https://www.comsol.com/support/learning-center>

Part number: CM020801

C o n t e n t s

Chapter 1: Introduction

About the Heat Transfer Module	24
Why Heat Transfer Is Important to Modeling	24
How the Heat Transfer Module Improves Your Modeling	25
The Heat Transfer Module Physics Interface Guide	25
Common Physics Interface and Feature Settings and Nodes	34
The Heat Transfer Module Study Capabilities	34
Additional Materials Database	39
Where Do I Access the Documentation and Application Libraries?	40
 Overview of the User's Guide	 43

Chapter 2: Notations

Symbols	50

Chapter 3: Modeling with the Heat Transfer Module

Heat Transfer Variables	66
Predefined Variables	66
Global Variables for Heat and Energy Balance	70
Domain Fluxes.	73
Out-of-Plane Domain Fluxes	76
Boundary Fluxes (Heat Transfer Interface)	77
Internal Boundary Heat Fluxes.	78
Internal Boundary Energy Fluxes	79
Domain Heat Sources	80
Boundary Heat Sources	81
Line and Point Heat Sources	81
Moist Air Variables	81

Moisture Transport Variables	84
Predefined Variables	84
Moist Air Properties	86
Domain Moisture Fluxes	88
Boundary Moisture Fluxes	89
Domain Moisture Source.	90
Mass Balance	90
 Using the Boundary Conditions for the Heat Transfer	
Interfaces	92
Temperature and Heat Flux Boundary Conditions	92
Overriding Mechanism for Heat Transfer Boundary Conditions	93
 Handling Frames in Heat Transfer	96
Heat Transfer Analysis with Moving Frames.	96
Material Density in Features Defined in the Material Frame.	101
 Heat Transfer Consistent and Inconsistent Stabilization	
Methods	103
Consistent Stabilization	103
Inconsistent Stabilization	104
 Heat Transfer and Fluid Flow Coupling	105
Coupling Based on Model Inputs	105
Adding Nonisothermal Flow Coupling in an Existing Model	106
Nonisothermal Flow and Conjugate Heat Transfer Multiphysics	
Interfaces	106
 Boundary Wall Temperature	108
 Solver Settings	113
Linearity Property of the Temperature Equation	113
Linear Solver	114
Nonlinear Solver	116
Time-Dependent Study Step	119
Guidelines for Solving Surface-to-Surface Radiation Problems	121
Guidelines for Solving Multiphysics Problems	121

Plotting and Evaluating Results in Layered Materials	124
Plotting Along and Through the Layered Material	124
Built-in Operators for Evaluation in the Layered Material	125
 Evaluating the Bulk Temperature	 129
Bulk Temperature Definition	129
Evaluation of the Bulk Temperature in 3D Models	130
Evaluation of the Bulk Temperature in 2D Models	130
 Ambient Properties	 133
Settings for the Ambient Properties.	133
Processing of ASHRAE Data	138
Ambient Data Interpolation	139
Ambient Variables and Conditions	139
Use of Ambient Data from the Features	148
 Modeling Heat Transfer by Radiation	 150
Interfaces and Features for Radiation in Transparent Media.	152
Surface-to-Surface Radiation and Topology Consistency	153
Interfaces and Features for Radiation in Participating Media.	154
 Specifying the Temperature at a Flow Inlet	 157
Using the Temperature Condition	157
Using the Inflow Condition	157
Temperature Contribution to the Inflow Boundary Condition.	159
Pressure Contribution to the Inflow Boundary Condition	160
Choosing Between the Temperature and the Inflow Conditions	160
 Heat Part Library	 161
Heat Part Library Contents	161
Heat Sink — Parameterized Fin Types Part	164
Heat Sink — Dissimilar Border Pins Part.	167
Heat Sink — Pin Fins Part	167
Heat Sink — Straight Fins Part.	167

Chapter 4: Theory for the Heat Transfer Module

Foundations of the General Heat Transfer Equation 171

Thermodynamic Description of Heat Transfer. 171

The Physical Mechanisms Underlying Heat Transfer. 176

The Heat Balance Equation 177

Consistency with Mass and Momentum Conservation Laws. 180

Theory for Heat Transfer in Solids 182

Theory for Heat Transfer in Fluids 183

Theory for the Inflow Boundary Condition 184

Theory for Heat Transfer in Moist Air 186

Humidity 186

Saturation State 188

Moist Air Properties 188

Theory for Bioheat Transfer 193

The Bioheat Equation 193

Damaged Tissue 193

Theory for Heat Transfer in Porous Media 197

When Should Local Thermal Equilibrium and Nonequilibrium Be
 Considered? 198

Local Thermal Equilibrium 199

Local Thermal Nonequilibrium 201

Packed Bed of Pellets 203

Theory for Heat Transfer in Moist Porous Media 206

Theory for Heat Transfer with Phase Change 208

Heat Transfer with Phase Change 208

Arbitrary Lagrangian-Eulerian Method 208

Apparent Heat Capacity Method 210

Theory for Heat Transfer in Building Materials	214
Theory for Harmonic Heat Transfer	216
Theory for Lumped Isothermal Domain	218
Theory for Heat Transfer in Thin Structures	221
Modeling Layered Materials	221
Theoretical Background of the Different Formulations	226
Thin Layer	231
Thin Film	235
Fracture	237
Thin Rod	238
Theory for the Lumped Thermal System Interface	239
Temperature Differences and Heat Rates in the Lumped Thermal System	239
Theory for the Conductive Thermal Resistor Component	241
Theory for the Radiative Thermal Resistor Component	244
Theory for the Thermal Capacitor and Thermal Mass Components	247
Theory for the Heat Pipe Component.	248
Theory for the Thermoelectric Module Component	250
Theory for Cell Periodicity	254
Theory for Thermal Connection	257
Conductive Thermal Resistor	257
Thermal Capacitor	260
Theory for Surface-to-Surface Radiation	263
Deriving the Radiative Heat Flux for Opaque Surfaces.	263
Deriving the Radiative Heat Flux for Semitransparent Surfaces	264
Wavelength Dependence of Surface Properties	266
Directional Dependence of Surface Properties	273
The Radiosity Method for Diffuse-Gray Surfaces	273
The Radiosity Method for Diffuse-Spectral Surfaces.	276
View Factor Evaluation	279

Theory for Orbital Thermal Loads	284
Coordinate Systems	284
Defining the Position of the Spacecraft Over Time	285
Orienting the Spacecraft	287
Modeling the Planet	289
Modeling of Fast Tumbling	290
 Theory for Radiation in Participating Media	 291
Radiation and Participating Media Interactions	291
Radiative Transfer Equation	292
Boundary Condition for the Radiative Transfer Equation	294
Heat Transfer Equation in Participating Media	296
Discrete Ordinates Method (DOM)	297
Discrete Ordinates Method Implementation in 2D	299
Discrete Ordinates Method Implementation in 2D Axisymmetry.	300
PI Approximation Theory	301
Radiation in Absorbing-Scattering Media Theory	305
Polychromatic Radiation	306
Radiative Beam in Absorbing Media Theory.	307
Rosseland Approximation Theory	308
 Theory for Moisture Transport	 310
Theory for Moisture Transport in Air	310
Theory for Moisture Transport in Building Materials	312
 Theory for Moisture Transport in Porous Media	 315
Equilibrium Moisture Transport	317
Nonequilibrium Moisture Transport	320
Capillary Pressure Models	322
 Theory for the Heat Transfer Multiphysics Couplings	 324
Theory for the Nonisothermal Flow and Conjugate Heat Transfer	
Interfaces	324
Theory for the Moisture Flow Interface	333
Theory for the Thermoelectric Effect Interface	339
Theory for the Building Materials Version of the Heat and Moisture	
Transport Interface	342
Theory for the Moist Air Version of the Heat and Moisture	

Transport Interface	342
Theory for the Moist Porous Media Version of the Heat and Moisture Transport Interface	343
Theory for the Heat and Moisture Flow Interfaces	344
Theory for the Electromagnetic Heating Interfaces	344
Theory for the Thermal Stress Interface	345
Theory for Thermal Contact	346
Theory for the Thermal Contact Feature	346
Out-of-Plane Heat Transfer	351
Equation Formulation	351
Convective Heat Transfer Correlations	354
Defining the Heat Transfer Coefficients	355
Nature of the Flow — The Grashof Number	356
Heat Transfer Coefficients — External Natural Convection	358
Heat Transfer Coefficients — Internal Natural Convection	365
Heat Transfer Coefficients — External Forced Convection	366
Heat Transfer Coefficients — Internal Forced Convection	369
Using the Heat and Mass Transfer Analogy for the Evaluation of Moisture Transfer Coefficients	370
Nucleate Pool Boiling Correlation	371
Rohsenow Correlation for Pool Nucleate Boiling	371
Critical Heat Flux for Nucleate Boiling.	372
Equivalent Thermal Conductivity Correlations	373
Horizontal Cavity With Bottom Heating	374
Vertical Cavity With Sidewall Heating	374
Temperature Dependence of Surface Tension	376
Heat Flux and Heat Balance	377
Total Heat Flux and Energy Flux	377
Heat and Energy Balance	378

Frames for the Heat Transfer Equations	381
Material and Spatial Frames	381
Conversion Between Material and Spatial Frames	383
References	387

Chapter 5: The Heat Transfer Module Interfaces

The Heat Transfer Interfaces	395
The Conjugate Heat Transfer Interfaces	398
The Radiation Interfaces	400
The Electromagnetic Heating Interfaces	404
The Thin Structures Interfaces	406
The Heat and Moisture Transport Interfaces	408
The Heat and Moisture Flow Interfaces	410
The Moisture Transport Interfaces	412
The Moisture Flow Interfaces	415
The Porous Media Interfaces	417
The Metal Processing Interfaces	419
The Curing Interfaces	421
About the Heat Transfer Interfaces	422
Space Dimensions	422
Study Types.	423
Versions of the Heat Transfer Interface	423

Benefits of the Different Heat Transfer Interfaces	424
Versions of the Heat Transfer in Shells Interface	424
Benefits of the Different Heat Transfer in Shells Interfaces	425
Layer and Interface Selection Tools	425
Using the Extra Dimension Coordinates	429
Versions of the Moisture Transport Interface	429
Benefits of the Different Moisture Transport Interfaces	430
Additional Physics Options	431
Settings for the Heat Transfer Interface	432
Feature Nodes for the Heat Transfer Interface	434
Settings for the Heat Transfer in Shells Interface	437
Settings for the Moisture Transport Interface	442
Feature Nodes for the Moisture Transport Interface	444
 The Heat Transfer in Solids Interface	 446
 The Heat Transfer in Fluids Interface	 447
 The Heat Transfer in Solids and Fluids Interface	 448
 The Heat Transfer in Porous Media Interface	 449
 The Local Thermal Nonequilibrium Interface	 450
 The Heat Transfer in Packed Beds Interface	 452
 The Heat Transfer in Moist Porous Media Interface	 454
 The Heat Transfer in Building Materials Interface	 455
 The Heat Transfer in Moist Air Interface	 456
 The Bioheat Transfer Interface	 457
 The Heat Transfer in Shells Interface	 458
Feature Nodes for the Heat Transfer in Shells Interface	460

The Heat Transfer in Films Interface	463
Feature Nodes for the Heat Transfer in Films Interface	465
The Heat Transfer in Fractures Interface	468
Feature Nodes for the Heat Transfer in Fractures Interface	470
The Lumped Thermal System Interface	473
Feature Nodes for the Lumped Thermal System Interface	474
The Surface-to-Surface Radiation Interface	476
Settings for the Surface-to-Surface Radiation Interface.	476
Feature Nodes for the Surface-to-Surface Radiation Interface	485
Predefined Functions for the Surface-to-Surface Radiation Interface . . .	485
The Orbital Thermal Loads Interface	488
Settings for the Orbital Thermal Loads Interface	489
Feature Nodes for the Orbital Thermal Loads Interface	489
Predefined Functions for the Orbital Thermal Loads Interface.	489
The Radiation in Participating Media Interface	490
Settings for the Radiation in Participating Media Interface	490
Feature Nodes for the Radiation in Participating Media Interface.	495
Predefined Functions for the Radiation in Participating Media Interface.	496
The Radiation in Absorbing-Scattering Media Interface	499
Settings for the Radiation in Absorbing-Scattering Media Interface . . .	499
Feature Nodes for the Radiation in Absorbing-Scattering Media Interface.	503
Predefined Functions for the Radiation in Absorbing-Scattering Media Interface.	505
The Radiative Beam in Absorbing Media Interface	506
Settings for the Radiative Beam in Absorbing Media Interface	506
Feature Nodes for the Radiative Beam in Absorbing Media Interface . . .	507

The Moisture Transport in Air Interface	509
The Equilibrium Moisture Transport in Porous Media Interface	510
The Nonequilibrium Moisture Transport in Porous Media Interface	511
The Moisture Transport in Building Materials Interface	513
The Moisture Transport in Free and Porous Media Interface	514
The Moisture Transport in Solids Interface	516

Chapter 6: The Heat Transfer Features

Domain Features	518
Absorbing Medium (Radiative Beam in Absorbing Medium Interface) . .	519
Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface)	521
Battery Layers	526
Bioheat	527
Biological Tissue	529
Building Material	531
Cell Periodicity	536
Convectively Enhanced Conductivity	539
Cross Section	541
Fluence Rate Calculation (Surface-to-Surface Radiation Interface) . . .	542
Fluid	543
Fluid (Porous Medium)	548
Geothermal Heating	553
Heat Source	554
Heat Source (Pellets)	557
Immobile Fluids (Porous Medium)	558
Initial Values	560
Initial Values (Pellets)	561

Initial Values (Radiative Beam in Absorbing Medium Interface)	562
Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	562
Irreversible Transformation	565
Isothermal Domain	568
Liquid Water (Moist Porous Medium)	570
Moist Air (Heat Transfer Interface)	571
Moist Air (Moist Porous Medium)	574
Moist Porous Medium	574
Opacity (Surface-to-Surface Radiation and Orbital Thermal Loads Interfaces)	578
Optically Thick Participating Medium	580
Out-of-Plane Heat Flux	582
Out-of-Plane Radiation	584
Participating Medium (Radiation in Participating Media Interface)	586
Pellets (Porous Medium)	592
Pellet-Fluid Interface (Porous Medium)	594
Phase Change Material	595
Porous Matrix (Porous Medium, Moist Porous Medium)	600
Porous Medium	604
Pressure Work	610
Radiative Source	611
Shape Memory Alloy	613
Solid	617
Solid with Translational Motion	621
Thermal Damage	622
Thermal Dispersion	626
Thermoelastic Damping	628
Thickness	629
Translational Motion	630
Viscous Dissipation	631
Boundary Features	633
Boundary Heat Source	635
Boundary Pair	638
Continuity	639
Continuity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	641

Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	642
Deposited Beam Power	642
Diffuse Mirror (Surface-to-Surface Radiation Interface)	644
Diffuse Surface (Surface-to-Surface Radiation Interface)	645
External Temperature (Thin Layer, Thin Film, Fracture)	653
Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)	655
Harmonic Perturbation	659
Heat Flux.	660
Heat Source (Heat Transfer in Shells Interface)	665
Heat Source (Thin Layer, Thin Film, Fracture).	667
Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	670
Incident Intensity (Radiative Beam in Absorbing Medium Interface) . . .	673
Inflow	675
Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	676
Initial Values (Surface-to-Surface Radiation Interface)	678
Isothermal Domain Interface	680
Layer Opacity (Surface-to-Surface Radiation Interface)	684
Line Heat Source on Axis	686
Local Thermal Nonequilibrium Boundary.	686
Lumped System Connector	687
Opaque Surface (Surface-to-Surface Radiation Interface).	688
Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	692
Opaque Surface (Radiative Beam in Absorbing Medium Interface) . . .	695
Open Boundary	696
Outflow	698
Periodic Condition (Heat Transfer Interface)	698
Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	700
Phase Change Interface	700
Phase Change Interface, Exterior.	704
Prescribed Radiosity (Surface-to-Surface Radiation Interface)	707
Radiation Group (Surface-to-Surface Radiation Interface)	711
Semitransparent Surface (Radiation in Participating Media and	

Radiation in Absorbing-Scattering Media Interfaces)	714
Semitransparent Surface (Surface-to-Surface Radiation Interface).	719
Surface-to-Ambient Radiation (Heat Transfer Interface)	723
Specular Mirror (Surface-to-Surface Radiation Interface)	725
Symmetry (Heat Transfer Interface).	726
Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)	726
Temperature	727
Thermal Contact	729
Thermal Insulation	734
Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)	735
Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)	740
Transparent Surface (Radiative Beam in Absorbing Medium Interface).	745
Boundary Interface Features	747
Deposited Beam Power, Interface (Heat Transfer in Shells Interface)	747
Heat Flux, Interface (Heat Transfer in Shells Interface)	750
Heat Source, Interface (Heat Transfer in Shells Interface)	753
Lumped System Connector, Interface (Heat Transfer in Shells Interface)	756
Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)	758
Temperature, Interface (Heat Transfer in Shells Interface)	761
Thermal Contact, Interface (Heat Transfer in Shells Interface).	763
Thermal Insulation, Interface (Heat Transfer in Shells Interface)	766
Edge Features	769
Heat Flux (Heat Transfer in Shells Interface)	769
Heat Flux (Thin Layer, Thin Film, Fracture)	772
Heat Source (Heat Transfer in Shells Interface)	775
Line Heat Source.	778
Lumped System Connector (Heat Transfer in Shells Interface)	780
Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)	782
Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	784

Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	786
Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	788
Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)	790
Thermal Insulation, Edge	792
Thin Rod	793
Point Features	795
Point Heat Flux (Thin Rod)	795
Point Heat Source	796
Point Heat Source on Axis	797
Surface-to-Ambient Radiation (Thin Rod)	798
Temperature (Thin Rod)	800
Global Features	802
Thermal Connection	803
Events Timeline	807
External Radiation Source	809
Generate Events Interface	815
Ground Pointing	817
Implicit Event	818
Orbital Parameters	819
Planet Properties	821
Spacecraft Axes	823
Spacecraft Orientation.	824
Sun Properties.	825
Symmetry for Surface-to-Surface Radiation	827
Thermal Connection, Interface-Interface	831
Conductive Thermal Resistor	837
Convective Thermal Resistor	841
External Terminal	843
Heat Pipe.	844
Heat Rate	848
Heat Rate Source	850
Nucleate Boiling Heat Rate	851
Radiative Heat Rate	852

Radiative Thermal Resistor	854
Subsystem Definition	856
Subsystem Instance	856
Temperature	857
Thermal Capacitor	857
Thermal Mass	859
Thermoelectric Module	860

Chapter 7: The Moisture Transport Features

Domain Features	868
Building Material	868
Hygroscopic Porous Medium	871
Initial Values	875
Liquid Water (Hygroscopic Porous Medium)	876
Liquid Water Source	879
Moist Air (Hygroscopic Porous Medium).	880
Moist Air (Moisture Transport Interface).	882
Moisture Source	885
Porous Matrix (Hygroscopic Porous Medium).	886
Turbulent Mixing	888
Vapor Source	889
 Boundary Features	 891
Continuity	891
Free-Porous Interface	893
Inflow	894
Insulation.	896
Liquid Water Content.	898
Liquid Water Flux	898
Moist Surface	899
Moisture Content	901
Moisture Flux	904
Nonequilibrium Boundary	906
Open Boundary	906
Outflow	909

Periodic Condition	910
Porous Interface	912
Porous-Porous Interface	913
Symmetry	914
Thin Moisture Barrier	915
Vapor Content	916
Vapor Flux	918
Wet Surface	920
Global Features	922
Gravity	922

Chapter 8: Multiphysics Interfaces

The Nonisothermal Flow and Conjugate Heat Transfer	
Interfaces	926
Advantages of Using the Multiphysics Interfaces	926
The Nonisothermal Flow, Laminar Flow and Turbulent Flow	
Interfaces	927
The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow	
Interfaces	928
Settings for Physics Interfaces and Coupling Features	929
Coupling Feature.	930
Physics Interface Features	931
Preset Studies	933
The Heat Transfer with Surface-to-Surface Radiation Interface	
934	
The Heat Transfer with Surface-to-Surface Radiation Multiphysics	
Interface.	934
Physics Interface Features	936
Coupling Feature.	936
The Heat Transfer with Orbital Thermal Load Interface	937
The Heat Transfer with Orbital Thermal Loads Multiphysics Interface. .	937
Physics Interface Features	938

Coupling Feature.	939
The Heat Transfer with Radiation in Participating Media Interface	940
The Heat Transfer with Radiation in Participating Media Multiphysics Interface.	940
Physics Interface Features	941
Coupling Feature.	942
The Heat Transfer with Radiation in Absorbing-Scattering Media Interface	943
The Heat Transfer with Radiation in Absorbing-Scattering Media Multiphysics Interface.	943
Physics Interface Features	944
Coupling Feature.	945
The Heat Transfer with Radiative Beam in Absorbing Media Interface	946
The Heat Transfer with Radiative Beam in Absorbing Media Multiphysics Interface.	946
Physics Interface Features	947
Coupling Feature.	947
The Thermoelectric Effect Interface	948
About The Thermoelectric Effect Interface	948
Settings for Physics Interfaces and Coupling Features	949
Coupling Features	950
Physics Interface Features	951
The Heat and Moisture Transport Interfaces	952
The Heat and Moisture Transport Multiphysics Interfaces	952
The Moist Air Version of the Heat and Moisture Transport Multiphysics Interface.	953
The Moist Porous Media Version of the Heat and Moisture Transport Multiphysics Interface.	954
The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface.	956
Physics Interface Features	958

Coupling Feature.	958
The Moisture Flow Interfaces	959
The Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces	959
Physics Interface Features for the Laminar and Turbulent Flow Versions.	962
The Moisture Flow, Brinkman Equations Multiphysics Interface	964
Physics Interface Features for the Brinkman Equations Version	966
Preset Studies	966
Coupling Feature.	966
The Heat and Moisture Flow Interfaces	967
The Heat and Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces	967
Physics Interface Features for the Laminar and Turbulent Flow Versions.	971
The Heat and Moisture Flow, Brinkman Equations Interface	973
Physics Interface Features for the Brinkman Equations Version	976
Coupling Features	976
The Joule Heating Interface	977
The Joule Heating Interface	977
Coupling Feature.	977
The Laser Heating Interface	978
The Laser Heating Interface.	978
Coupling Feature.	978
The Induction Heating Interface	979
The Induction Heating Interface	979
Coupling Feature.	979
The Microwave Heating Interface	980
The Microwave Heating Interface	980
Coupling Feature.	980

Chapter 9: Multiphysics Couplings

Global Multiphysics Couplings	983
Thermal Connection, Layered Shell, Surfaces	983
Thermal Connection, Nonlayered Shell	985
 Domain Multiphysics Couplings	 988
Electromagnetic Heating	988
Heat and Moisture	989
Heat Transfer with Radiation in Participating Media.	992
Heat Transfer with Radiation in Absorbing-Scattering Media	994
Heat Transfer with Radiative Beam in Absorbing Media	995
Moisture Flow	996
Nonisothermal Flow	1000
Thermal Expansion	1007
Thermoelectric Effect	1007
 Boundary Multiphysics Couplings	 1011
Electromagnetic Heating, Layered Shell	1011
Heat Transfer with Surface-to-Surface Radiation	1013
Thermal Expansion, Layered	1015
Marangoni Effect	1015
Thermal Connection, Layered Shell, Edges	1018
Thermoelectric Effect, Layered Shell	1019
 Index	 1023

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 Application Library.

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 Application Library, 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 <i>COMSOL Multiphysics Reference Manual</i>: • 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
Equilibrium Moisture Transport in Porous Media		mt	all dimensions	stationary; time dependent
Nonequilibrium Moisture Transport in Porous Media		mt	all dimensions	stationary; time dependent
Moisture Transport in Building Materials		mt	all dimensions	stationary; time dependent
Moisture Transport in Free and Porous Media		mt	all dimensions	stationary; time dependent
Moisture Transport in Solids		mts	all dimensions	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
 Moisture Flow				
Laminar Flow ⁽²⁾		—	all dimensions	stationary; time dependent
 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 - ϵ ⁽²⁾		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, SST ⁽²⁾		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Turbulent Flow, k- ϵ		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Turbulent Flow, SST		spf	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
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
 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, SST ⁽²⁾		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
Turbulent Flow, Low Re k- ϵ ⁽²⁾		—	3D, 2D, 2D axisymmetric	stationary with initialization; transient with initialization
 Heat Transfer				
Heat Transfer in Solids ⁽¹⁾		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain; thermal perturbation, eigenfrequency

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
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
 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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
Turbulent Flow, SST ⁽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 Orbital Thermal Loads ⁽²⁾		—	3D	time dependent
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
Surface-to-Surface Radiation		rad	all dimensions	stationary; time dependent
Orbital Thermal Loads		otl	3D	time dependent
Radiation in Participating Media		rpm	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
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				
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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE PRESET STUDY TYPE
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
Turbulent Flow, k - ϵ ⁽²⁾		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF; one-way MF; time dependent, one-way MF; thermal perturbation, frequency domain

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

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

Common Physics Interface and Feature Settings and Nodes

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

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

	In the <i>COMSOL Multiphysics Reference Manual</i> 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 <i>COMSOL Multiphysics Reference Manual</i>

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	√	√				
Equilibrium Moisture Transport in Porous Media	mt	ϕ	√	√				
Nonequilibrium Moisture Transport in Porous Media	mt	ϕ, s_l	√	√				
Moisture Transport in Building Materials	mt	ϕ	√	√				
Moisture Transport in Free and Porous Media	mt	ϕ, s_l	√	√				
Moisture Transport in Solids	mts	ϕ, p_m	√	√				
CHEMICAL SPECIES TRANSPORT>MOISTURE FLOW								
Laminar Flow ⁽²⁾	—	u, p, ϕ	√	√				
Turbulent flow, Algebraic yPlus ⁽²⁾	—	$u, p, G,$ $yPlus, \phi$					√	√
Turbulent flow, L-VEL ⁽²⁾	—	$u, p, G,$ $uPlus, \phi$					√	√
Turbulent Flow, k-ε ⁽²⁾	—	u, p, k, ep, ϕ	√	√				
Turbulent Flow, SST ⁽²⁾	—	$u, p, k, om,$ G, ϕ					√	√
Turbulent Flow, Low Re k-ε ⁽²⁾	—	$u, p, k, ep, G,$ ϕ					√	√
Porous Media Flow ⁽²⁾	—	u, p, ϕ	√	√				

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
FLUID FLOW>SINGLE-PHASE FLOW								
Laminar Flow	spf	u, p	√	√				
Turbulent flow, Algebraic $yPlus$	spf	$u, p, G, yPlus$					√	√
Turbulent flow, L-VEL	spf	$u, p, G, uPlus$					√	√
Turbulent Flow, $k-\epsilon$	spf	u, p, k, ep	√	√				
Turbulent Flow, SST	spf	u, p, k, om, G					√	√
Turbulent Flow, Low Re $k-\epsilon$	spf	u, p, k, ep, G					√	√
FLUID FLOW>NONISOTHERMAL FLOW								
Laminar Flow ⁽²⁾	—	u, p, T	√	√				
Turbulent flow, Algebraic $yPlus$ ⁽²⁾	—	$u, p, G, yPlus, T$					√	√
Turbulent flow, L-VEL ⁽²⁾	—	$u, p, G, uPlus, T$					√	√
Turbulent Flow, $k-\epsilon$ ⁽²⁾	—	u, p, k, ep, T	√	√				
Turbulent Flow, SST ⁽²⁾	—	u, p, k, om, G, T					√	√
Turbulent Flow, Low Re $k-\epsilon$ ⁽²⁾	—	u, p, k, ep, G, T					√	√
HEAT TRANSFER								
Heat Transfer in Solids	ht	T	√	√	√	√		
Heat Transfer in Fluids	ht	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 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, γ Plus, T					√	√
Turbulent flow, L-VEL ⁽²⁾	—	u, p, G, γ Plus, T					√	√
Turbulent Flow, k - ϵ ⁽²⁾	—	u, p, k, ϵ, T	√	√				
Turbulent Flow, SST ⁽²⁾	—	u, p, k, ω, G, T					√	√
Turbulent Flow, Low Re k - ϵ ⁽²⁾	—	u, p, k, ϵ, G, T					√	√
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	√	√				

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
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								
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⁾	—	$u, p, G, uPlus, T, \phi$					✓	✓

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
Turbulent Flow, k- ϵ ⁽²⁾	—	u, p, k, ep, T, ϕ	√	√				
Turbulent Flow, SST ⁽²⁾	—	u, p, k, om, G, T, ϕ					√	√
Turbulent Flow, Low Re k- ϵ ⁽²⁾	—	u, p, k, ep, G, T, ϕ					√	√
Porous Media Flow ⁽²⁾	—	u, p, T, ϕ	√	√				
Heat Transfer in Porous Media	ht	T	√	√	√	√		
Local Thermal Nonequilibrium	ht	T	√	√	√	√		
Heat Transfer in Packed Beds	ht	T	√	√	√	√		
Bioheat Transfer	ht	T	√	√	√	√		
Thermoelectric Effect ⁽²⁾	—	T	√	√	√	√		

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

² Multiphysics interfaces.

Additional Materials Database

The Heat Transfer Module includes additional material databases:

- The Liquids and Gases material database provides temperature-dependent fluid dynamic and thermal properties (dynamic viscosity, heat capacity at constant pressure, density, and thermal conductivity).
- The Bioheat material database contains materials used with the Bioheat Transfer interface. In addition to thermal properties (heat capacity at constant pressure,

density, and thermal conductivity), properties for thermal damage analysis of biological tissues are provided.

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



For detailed information, see [Liquids and Gases Material 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 online 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 on 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	To open the Help window: • In the Model Builder, Application Builder, or Physics Builder, click a node or window and then press F1. • On any toolbar (for example, Home, Definitions, or Geometry), hover the mouse over a button (for example, Add Physics or Build All) and then press F1. • From the File menu, click Help (). • In the upper-right corner of the COMSOL Desktop, click the Help () button.
Mac Linux	To open the Help window: • In the Model Builder or Physics Builder, click a node or window and then press F1. • In the main toolbar, click the Help () button. • From the main menu, select Help>Help.

Opening the Documentation Window

Mac Linux	To open the Documentation window: • Press Ctrl+F1. • In 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.

Opening the Application Libraries Window

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

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 will be 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/product-update
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/videos
Learning Center	www.comsol.com/support/learning-center
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

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



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

TABLE OF CONTENTS, NOTATIONS, AND INDEX

To help you navigate through this guide, see the [Contents](#), [Notations](#), and [Index](#).

MODELING WITH THE HEAT TRANSFER MODULE

The [Modeling with the Heat Transfer Module](#) chapter includes the following topics:

- [Heat Transfer Variables](#)
- [Moisture Transport Variables](#)
- [Using the Boundary Conditions for the Heat Transfer Interfaces](#)
- [Handling Frames in Heat Transfer](#)
- [Heat Transfer Consistent and Inconsistent Stabilization Methods](#)
- [Heat Transfer and Fluid Flow Coupling](#)
- [Boundary Wall Temperature](#)
- [Solver Settings](#)
- [Plotting and Evaluating Results in Layered Materials](#)
- [Ambient Properties](#)
- [Modeling Heat Transfer by Radiation](#)
- [Heat Part Library](#)

THEORY FOR THE HEAT TRANSFER MODULE

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

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

- [Theory for Heat Transfer in Solids](#)
- [Theory for Heat Transfer in Fluids](#)
- [Theory for Bioheat Transfer](#)
- [Theory for Heat Transfer in Porous Media](#)
- [Theory for Heat Transfer with Phase Change](#)
- [Theory for Heat Transfer in Building Materials](#)
- [Theory for Harmonic Heat Transfer](#)
- [Theory for Lumped Isothermal Domain](#)
- [Theory for Heat Transfer in Thin Structures](#)
- [Theory for the Lumped Thermal System Interface](#)
- [Theory for Surface-to-Surface Radiation](#)
- [Theory for Radiation in Participating Media](#)
- [Theory for Moisture Transport](#)

Then the theory related to multiphysics interfaces is described in the section [Theory for the Heat Transfer Multiphysics Couplings](#).

Finally, topics related to specific features or variables are treated in the sections [Theory for Thermal Contact](#), [Theory for Heat Transfer in Moist Air](#), [Out-of-Plane Heat Transfer](#), [Convective Heat Transfer Correlations](#), [Equivalent Thermal Conductivity Correlations](#), [Temperature Dependence of Surface Tension](#), [Heat Flux and Heat Balance](#), and [Frames for the Heat Transfer Equations](#).

THE HEAT TRANSFER MODULE INTERFACES

The chapter [The Heat Transfer Module Interfaces](#) describes the main Heat Transfer interface (ht) that forms the backbone for all the fundamental interfaces in this module, and the other interfaces (Heat Transfer in Shells (htlsh), Radiation in Participating Media (rpm), Radiation in Absorbing-Scattering Media (rasm), Radiative Beam in Absorbing Media (rbam), Surface-to-Surface Radiation (rad), and Moisture Transport (mt)).

The sections [The Heat Transfer in Solids Interface](#), [The Heat Transfer in Fluids Interface](#), and [The Heat Transfer in Solids and Fluids Interface](#) discuss modeling heat transfer in solids and fluids.

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

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

The section [The Bioheat Transfer Interface](#) discusses modeling heat transfer within biological tissue using the Bioheat Transfer interface.

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

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

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

Finally, the sections [The Moisture Transport in Air Interface](#), [The Equilibrium Moisture Transport in Porous Media Interface](#), [The Nonequilibrium Moisture Transport in Porous Media Interface](#), [The Moisture Transport in Building Materials Interface](#), [The Moisture Transport in Free and Porous Media Interface](#), and [The Moisture Transport in Solids Interface](#) describe the modeling of moisture transfer in a porous medium through moisture storage or liquid water saturation, 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 [The Moisture Transport Features](#) chapter describes the [Domain Features](#), [Boundary Features](#), and [Global Features](#) available with the Moisture Transport interface.

THE HEAT TRANSFER MULTIPHYSICS INTERFACES

The [Multiphysics Interfaces](#) chapter describes the predefined multiphysics interfaces.

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

The section [The Heat Transfer with Surface-to-Surface Radiation Interface](#) describes the predefined multiphysics interface used to model heat transfer by conduction, convection, and radiation in a transparent media.

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

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

The section [The Thermoelectric Effect Interface](#) describes the predefined multiphysics interface used to model the Peltier-Seebeck-Thomson effect.

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

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

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

THE HEAT TRANSFER MULTIPHYSICS COUPLINGS

The [Multiphysics Couplings](#) chapter describes the [Domain Multiphysics Couplings](#) and the [Boundary Multiphysics Couplings](#) available with the predefined multiphysics interfaces.

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
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
$C_{p, v}$	J/(kg·K)	Specific heat capacity at constant pressure, water vapor

NOTATION	SI UNIT	DESCRIPTION
$C_{p, s}$	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_{pe}	m	Average pellets diameter
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
$\mathbf{e}$	m	Beam orientation
E	J/kg	Internal energy
E	Pa	Young's modulus
E_0	J/kg	Total internal energy
E_{Ω}	J	Internal energy of a body

NOTATION	SI UNIT	DESCRIPTION
$e_{b,\lambda}(\lambda, T)$	W/m^2	Blackbody spectral emissive power
$e_b(T)$	W/m^2	Blackbody total emissive power
E_{contact}	Pa	Effective contact interface Young's modulus
E_d	Pa	Young's modulus, down contact surface
E_k	J/kg	Kinetic energy
ep	m^2/s^3	Turbulent dissipation rate
E_p	J/kg	Potential energy
E_u	Pa	Young's modulus, up contact surface
$\mathbf{e}_{\text{tot}}$	W/m^2	Total energy flux
$\mathbf{F}$	N/m^3	Body force vector
$\mathbf{F}$	dimensionless	Deformation gradient tensor
F_{amb}	dimensionless	Ambient view factor
$F_{\text{amb, d}}$	dimensionless	Ambient view factor, downside
$F_{\text{amb, u}}$	dimensionless	Ambient view factor, upside
g	m/s^2	Acceleration of gravity
G_{evap}	$\text{kg}/(\text{m}^3 \cdot \text{s})$	Evaporation source
g_{evap}	$\text{kg}/(\text{m}^2 \cdot \text{s})$	Evaporation flux
G	$\text{kg}/(\text{m}^3 \cdot \text{s})$	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_l	$\text{kg}/(\text{m}^3 \cdot \text{s})$	Liquid water source
g_{lc}	$\text{kg}/(\text{m}^2 \cdot \text{s})$	Liquid water capillary flux
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

NOTATION	SI UNIT	DESCRIPTION
Gr_L	dimensionless	Grashof number associated with characteristic length L
G_u	W/m^2	Surface irradiation, upside
G_v	$kg/(m^3 \cdot s)$	Water vapor source
g_w	$kg/(m^2 \cdot s)$	Vapor diffusive flux
H	m	Chimney height (heat transfer coefficient)
H	J/kg	Enthalpy
h	$W/(m^2 \cdot K)$	Gap conductance (thermal contact)
h	$W/(m^2 \cdot K)$	Heat transfer coefficient
H_0	J/kg	Total enthalpy
H_B	Pa	Brinell hardness
h_c	$W/(m^2 \cdot K)$	Constriction conductance
H_c	Pa	Microhardness
h_d	$W/(m^2 \cdot K)$	Out-of-plane heat transfer coefficient, downside
H_d	J/kg	Enthalpy, downside
H_{ext}	J/kg	External enthalpy
h_g	$W/(m^2 \cdot K)$	Parallel-plate gap gas conductance
h_m	m/s	Moisture transfer coefficient
h_r	$W/(m^2 \cdot K)$	Radiative conductance
H_{ref}	J/kg	Reference enthalpy
h_{sf}	$W/(m^2 \cdot K)$	Interstitial heat transfer coefficient
h_{th}	W/K	Thermal conductance (isothermal domain interface)
h_u	$W/(m^2 \cdot K)$	Out-of-plane heat transfer coefficient, upside
H_u	J/kg	Enthalpy, upside
h_z	$W/(m^2 \cdot K)$	Out-of-plane heat transfer coefficient, ID
$I(\Omega)$	$W/(m^2 \cdot sr)$	Radiative intensity traveling in direction Ω
$I_b(T)$	$W/(m^2 \cdot sr)$	Blackbody radiative intensity
I_{diff}	W/m^2	Diffuse irradiance
I_{ext}	$W/(m^2 \cdot sr)$	Incident radiative intensity
I_i	$W/(m^2 \cdot sr)$	Radiative intensity traveling in i th discrete direction
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

NOTATION	SI UNIT	DESCRIPTION
$I_{\text{sh,amb}}$	W/m^2	Clear sky noon diffuse horizontal irradiance
$I_{\text{sn,amb}}$	W/m^2	Clear sky noon beam normal irradiance
$i_{s_x}, i_{s_y}, i_{s_z}$	dimensionless	Solar source direction vector components
I_{wall}	$\text{W}/(\text{m}^2 \cdot \text{sr})$	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	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity
k	J/kg	Turbulent kinetic energy (turbulent nonisothermal flow)
k_{a}	$\text{W}/(\text{m} \cdot \text{K})$	Dry air thermal conductivity
k_{A}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of Austenite state
k_{B}	J/K	Stefan-Boltzmann constant
k_{bnd}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity in shell local coordinate system
k_{contact}	$\text{W}/(\text{m} \cdot \text{K})$	Harmonic mean of contacting surface conductivities
k_{d}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity, damaged tissue
k_{disp}	$\text{W}/(\text{m} \cdot \text{K})$	Dispersive thermal conductivity tensor
k_{eff}	$\text{W}/(\text{m} \cdot \text{K})$	Effective thermal conductivity
K_{evap}	l/s	Evaporation rate
k_{f}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity, fluid phase
k_{imf}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of immobile fluid in porous media
k_{gap}	$\text{W}/(\text{m} \cdot \text{K})$	Parallel-plate gap gas thermal conductivity
k_{imfi}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of immobile fluid i in porous media
k_{l}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of thin rod
k_{L}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of mobile fluid in porous media
k_{M}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of Martensite state
k_{R}	$\text{W}/(\text{m} \cdot \text{K})$	Rosseland radiative conductivity
k_{s}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity, solid phase
k_{si}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of solid material i in porous media and fractures
k_{s}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of thin layer
k_{si}	$\text{W}/(\text{m} \cdot \text{K})$	Thermal conductivity of layer i in thin layer
k_{T}	$\text{W}/(\text{m} \cdot \text{K})$	Turbulent thermal conductivity

NOTATION	SI UNIT	DESCRIPTION
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
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
l_{vG}	dimensionless	van Genuchten model constitutive constant
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
m_{vG}	dimensionless	van Genuchten model constitutive constant
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

NOTATION	SI UNIT	DESCRIPTION
om	l/s	Specific dissipation rate
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
P_b	W	Heat rate, boundary heat source
P_c	m	Cross sectional perimeter of domain
p_{ec}	Pa	Entry capillary pressure
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
$p_{v,eq}$	Pa	Equilibrium vapor 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

NOTATION	SI UNIT	DESCRIPTION
$q_{0, d}$	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
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	$W/(m^3 \cdot K)$	Production/absorption coefficient
Q_s	W/m^3	Heat source in solid phase
Q_s	W/m^3	Thin layer heat source
q_{sf}	$W/(m^3 \cdot K)$	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

NOTATION	SI UNIT	DESCRIPTION
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
r_h	m	Hydraulic radius
r_l	m	Rod radius
r_{pe}	m	Average pellets radius
R_s	J/(kg·K)	Specific gas constant
$R_{t,s}$	K·m ² /W	Surface thermal resistance
R_t	K/W	Thermal resistance
S_b	l/m	Specific surface area
s_d	m	Vapor diffusion equivalent air layer thickness
$\mathbf{S}$	N/m ²	Second Piola-Kirchhoff stress tensor
S	V/K	Seebeck coefficient
Sc_T	dimensionless	Turbulent Schmidt number
$\mathbf{S}_i$	dimensionless	Unit vector of discrete direction in space, i th component (angular space discretization)
s_l	dimensionless	Liquid water saturation
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

NOTATION	SI UNIT	DESCRIPTION
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
$T_{it,c}$		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
$\mathbf{u}_g$	m/s	Moist air velocity vector
$\mathbf{u}_l$	m/s	Liquid water velocity vector
$\mathbf{u}_p$	m/s	Porous velocity field
$\mathbf{u}_{trans}$	m/s	Translational motion velocity vector
v_{amb}	m/s	Wind velocity
V	V	Electric potential

NOTATION	SI UNIT	DESCRIPTION
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
$x_{\text{vap,amb}}$	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

NOTATION	SI UNIT	DESCRIPTION
β	dimensionless	Parallel-plate gap gas property parameter
β_R	l/m	Rosseland mean extinction coefficient
γ	dimensionless	Ratio of specific heats
γ_{Teq}	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
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	l/m	Absorption coefficient (radiation)
κ	m ²	Permeability (porous media)
κ_{rg}	dimensionless	Moist air relative permeability (porous media)
κ_{rl}	dimensionless	Liquid water relative permeability (porous media)
λ	W/(m·K)	Thermal conductivity (turbulent nonisothermal flow)

NOTATION	SI UNIT	DESCRIPTION
λ	m	Wavelength
Λ	m	Parallel-plate gap gas mean free path
λ_i	m	Wavelength band i endpoint
λ_{ijkl}	m	Fourth-order dispersivity tensor's component
λ_{lo}	m	Longitudinal dispersivity
λ_p	dimensionless	Pore size distribution index
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
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

NOTATION	SI UNIT	DESCRIPTION
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
$\phi_{w,u}$	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 th component

NOTATION	SI UNIT	DESCRIPTION
ω_v	dimensionless	Water vapor mass fraction
ξ	kg/m^3	Moisture storage capacity
ξ	dimensionless	Martensite state volume fraction

Modeling with the Heat Transfer Module

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

- [Heat Transfer Variables](#)
- [Moisture Transport Variables](#)
- [Using the Boundary Conditions for the Heat Transfer Interfaces](#)
- [Handling Frames in Heat Transfer](#)
- [Heat Transfer Consistent and Inconsistent Stabilization Methods](#)
- [Heat Transfer and Fluid Flow Coupling](#)
- [Boundary Wall Temperature](#)
- [Solver Settings](#)
- [Plotting and Evaluating Results in Layered Materials](#)
- [Evaluating the Bulk Temperature](#)
- [Ambient Properties](#)
- [Modeling Heat Transfer by Radiation](#)
- [Specifying the Temperature at a Flow Inlet](#)
- [Heat Part Library](#)

Heat Transfer Variables

In this section:

- [Predefined Variables](#)
- [Global Variables for Heat and Energy Balance](#)
- [Domain Fluxes](#)
- [Out-of-Plane Domain Fluxes](#)
- [Boundary Fluxes \(Heat Transfer Interface\)](#)
- [Internal Boundary Heat Fluxes](#)
- [Domain Heat Sources](#)
- [Boundary Heat Sources](#)
- [Line and Point Heat Sources](#)
- [Moist Air Variables](#)

Predefined Variables

This section lists some predefined variables that are available to evaluate heat fluxes, sources, and integral quantities used in heat and energy balance. All the variable names begin with the physics interface name (the prefix). By default the Heat Transfer interface prefix is `ht`, and the Heat Transfer in Shells interface prefix is `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		
dEiInt	Total Accumulated Heat Rate	Global
ntfluxInt	Total Net Heat Rate	Global
WstrInt	Total Stress Power	Global
heatBalance	Heat Balance	Global
Domain variables		
tflux	Total Heat Flux (Heat Transfer Interface)	Domains

TABLE 3-1: PREDEFINED HEAT BALANCE VARIABLES.

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

TABLE 3-2: PREDEFINED ENERGY BALANCE VARIABLES.

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

TABLE 3-3: PREDEFINED HEAT SOURCE VARIABLES.

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

TABLE 3-3: PREDEFINED HEAT SOURCE VARIABLES.

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

TABLE 3-4: OTHER PREDEFINED HEAT TRANSFER VARIABLES.

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
Global variables		
Tref	Reference temperature	Global
Domain variables		
alpha	Damage indicator (see Theory for Bioheat Transfer)	Domains
alphanecr	Instant tissue necrosis indicator (see Theory for Bioheat Transfer)	Domains
theta_d	Fraction of damage (see Theory for Bioheat Transfer)	Domains
theta_d_sm	Fraction of total damage (see Theory for Bioheat Transfer)	Domains
T_dp	Dew Point Temperature	Domains
T_eq	Equivalent Temperature	Domains
psat	Saturation Pressure	Domains
phi	Relative Humidity	Domains
Lv	Latent Heat of Evaporation	Domains
Boundary variables		
h	Heat transfer coefficient (see Convective Heat Transfer Correlations)	Exterior boundaries



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

Global Variables for Heat and Energy Balance

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

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

TOTAL NET HEAT RATE

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

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

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:

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

TOTAL STRESS POWER

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

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

TOTAL ACCUMULATED ENERGY RATE

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

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

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

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

TOTAL NET ENERGY RATE

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

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

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

TOTAL WORK SOURCE

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

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

where $\mathbf{K}$ is the viscous stress 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 4-209](#), the following equality between COMSOL Multiphysics variables holds:

$$\text{dEiInt} + \text{ntefluxInt} = \text{QInt} - \text{WstrInt}$$

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

$$\text{heatBalance} = \text{dEiInt} + \text{ntfluxInt} + \text{WstrInt} - \text{QInt}$$

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

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

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:

$$\int_{\partial\Omega_{\text{ext}}} \rho \mathbf{u} E_1 \cdot \mathbf{n} d\sigma - \int_{\partial\Omega_{\text{ext}}} k \nabla T \cdot \mathbf{n} d\sigma = Q_{\text{Int}}$$



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`.



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

ENERGY BALANCE

According to [Equation 4-210](#), the following equality between COMSOL Multiphysics predefined variables holds:

$$dEi0Int + ntefluxInt = QInt + WInt$$

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

$$energyBalance = dEi0Int + ntefluxInt - WInt - QInt$$

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

$$ntefluxInt = QInt + WInt$$

At steady state, and without any additional heat source (`QInt` equal to zero), or external forces, the energy balance reduces to the integral of the total net energy flux (conduction, convection, radiation, and viscous dissipation) over all boundaries of the flow, `ntefluxInt + WInt` therefore 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 <code>ht.ntefluxInt</code> or <code>ht.ntefluxInt</code> 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 <code>ht.ntefluxInt</code> . It is instead added to the variable <code>ht.sls1.ntefluxInt</code> , defined locally on the thin structure. Here, <code>sls1</code> is the tag of the Thin Layer node.
	The energy balance can be evaluated assuming constant material properties using the following variables: <code>energyBalance_cst</code> , <code>dEi0Int_cst</code> , <code>ntefluxInt_cst</code> , <code>WInt_cst</code> , and <code>QInt_cst</code> . The evaluation of these variables is faster and accurate for constant material properties or small temperature gradients.

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:

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

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

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

CONDUCTIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

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

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 4-162](#).

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.

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

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 as:

$$\text{teflux} = \text{thflux} + \text{dflux}$$

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

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

- In 2D:

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

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

- In 1D:

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

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:

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

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:

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

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

- In 1D:

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

Boundary Fluxes (Heat Transfer Interface)

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

NORMAL TOTAL HEAT FLUX (HEAT TRANSFER INTERFACE)

The variable `ntflux` is defined as:

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

NORMAL CONDUCTIVE HEAT FLUX (HEAT TRANSFER INTERFACE)

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

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

and, on interior boundaries, as:

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



[Frames for the Heat Transfer Equations](#) for a description of spatial and material frames.

NORMAL CONVECTIVE HEAT FLUX (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, q_0 , is a scalar quantity defined either in a general way or as a convective heat flux as follows:

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

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:

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

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:

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



All the boundary fluxes or boundary sources balance the normal conductive flux jump across the boundary. Hence the normal diffusive flux account for the radiative heat flux if present in the model.

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

The sum of the point heat sources is available in a variable called Q_{ptot} (SI unit: W).

Moist Air Variables

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

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

DEW POINT TEMPERATURE

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

The variable T_{dp} is defined in [Ref. 1](#) by:

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

where ϕ 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 p_{sat} .

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 an increase of the sensible heat and temperature.

The variable T_{eq} is approximated in [Ref. 1](#) by:

$$T_{eq} = T + \frac{\phi \cdot p_{sat}}{\gamma_{Teq}}$$

where ϕ is the [Relative Humidity](#), and γ_{Teq} (SI unit: Pa/K) is the psychrometer constant, defined in [Ref. 1](#) by:

$$\gamma_{Teq} = \frac{M_a p C_{p,a}}{M_v L_v}$$

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 p_{sat} .

These definitions are illustrated on [Figure 3-1](#).

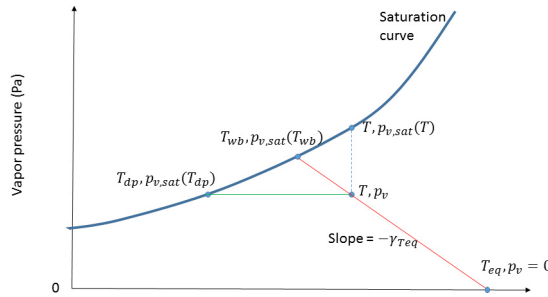


Figure 3-1: Relation between dry bulb, dew point, equivalent, and wet bulb temperatures.

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:

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

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:

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

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>dwcInt</code>	Total accumulated moisture rate	Global
<code>ntfluxInt</code>	Total net moisture rate	Global
<code>GInt</code>	Total moisture source	Global
<code>massBalance</code>	Mass balance	Global
<code>phi</code>	Relative humidity	Domains
<code>cv</code>	Vapor concentration	Domains
<code>cl</code>	Liquid water concentration	Domains
<code>cw</code>	Total moisture concentration	Domains
<code>ca</code>	Dry air 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>wcVar</code>	Moisture content	Domains
<code>omega_v</code>	Vapor mass fraction	Domains
<code>omega_moist</code>	Moisture mass fraction	Domains
<code>xvap</code>	Humidity ratio	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

TABLE 3-5: MOISTURE TRANSPORT PREDEFINED VARIABLES.

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
rho_moist	Density of moist air	Domains
mua	Viscosity of dry air	Domains
muv	Viscosity of water vapor	Domains
mu_moist	Viscosity of moist air	Domains
cflux	Convective moisture flux	Domains
dflux	Diffusive moisture flux	Domains
tflux	Total moisture flux	Domains
tfluxMag	Total moisture flux magnitude	Domains
gw	Vapor diffusive flux	Domains
glc	Capillary liquid flux	Domains
Gtot	Total moisture source	Domains
G1	Liquid water source	Domains
Gv	Vapor source	Domains
dwc	Total accumulated moisture rate density	Domains
h_m	Moisture transfer coefficient	Boundaries
ncflux	Normal convective moisture flux	Boundaries
ndflux	Normal diffusive moisture flux	Boundaries
ntflux	Normal total moisture flux	Boundaries
g_evap	Evaporation flux	Boundaries
q_evap	Latent heat source	Boundaries
q_evaptot	Total latent heat source on surfaces	Boundaries
gtot	Total moisture flux	Boundaries
uStefan_u	Stefan velocity, upside	Boundaries
uStefan_d	Stefan velocity, downside	Boundaries
ncflux_u	Normal convective moisture flux, upside	Interior boundaries
ncflux_d	Normal convective moisture flux, downside	Interior boundaries
ndflux_u	Normal diffusive moisture flux, upside	Interior boundaries
ndflux_d	Normal diffusive moisture flux, downside	Interior boundaries

TABLE 3-5: MOISTURE TRANSPORT PREDEFINED VARIABLES.

VARIABLE	NAME	GEOMETRIC ENTITY LEVEL
ntflux_u	Normal total moisture flux, upside	Interior boundaries
ntflux_d	Normal total moisture flux, downside	Interior boundaries

Moist Air Properties

PARTIAL AND SATURATION PRESSURES OF WATER VAPOR

The partial pressure of vapor 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.

HUMIDITY RATIO

The humidity ratio x_{vap} is obtained from the vapor and air concentrations c_v and c_a , the absolute pressure p_A and temperature T by:

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

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

The vapor and dry air concentrations satisfy:

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

VAPOR MASS FRACTION

The vapor mass fraction ω_v is obtained by:

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

LIQUID, VAPOR, AND MOISTURE CONCENTRATIONS

The total moisture concentration 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 $c_l=0$ in moist air domains.

CONCENTRATIONS AND CONTENTS

The liquid water and vapor contents w_{c_l} and w_{c_v} are obtained by multiplying the concentrations c_l and c_v by the molar mass of water M_v :

$$w_{c\_l} = M_v \cdot c_l$$

$$w_{c\_v} = M_v \cdot c_v$$

MOISTURE CONTENT

The moisture content w_{cVar} is expressed in different ways depending on the domain type:

- In moist air:

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

- In hygroscopic porous media:

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

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

LATENT HEAT OF EVAPORATION

The variable 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:

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

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

TOTAL MOISTURE FLUX

The variable `tflux` is defined as:

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

TOTAL MOISTURE FLUX MAGNITUDE

The variable `tfluxMag` is defined as:

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

Boundary Moisture Fluxes

NORMAL CONVECTIVE MOISTURE FLUX

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

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

INTERNAL NORMAL CONVECTIVE MOISTURE FLUX, UPSIDE

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

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

INTERNAL NORMAL CONVECTIVE MOISTURE FLUX, DOWNSIDE

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

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

NORMAL DIFFUSIVE MOISTURE FLUX

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

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

INTERNAL NORMAL DIFFUSIVE MOISTURE FLUX, UPSIDE

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

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

INTERNAL NORMAL DIFFUSIVE MOISTURE FLUX, DOWNSIDE

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

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

NORMAL TOTAL MOISTURE FLUX

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

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

INTERNAL NORMAL TOTAL MOISTURE FLUX, UPSIDE

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

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

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`.

Mass Balance

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

$$\text{massBalance} = \text{dwcInt} + \text{ntfluxInt} - \text{GInt}$$

where:

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

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

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

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

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

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

Using the Boundary Conditions for the Heat Transfer Interfaces

In this section:

- [Temperature and Heat Flux Boundary Conditions](#)
- [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.

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

When there is a boundary condition *A* above a boundary condition *B* in the model tree and both conditions apply to the same boundary, use [Table 3-6](#) to determine if *A* is overridden by *B* or not:

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

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

Example

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

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

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

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

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



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

Handling Frames in Heat Transfer

In this section:

- [Heat Transfer Analysis with Moving Frames](#)
- [Material Density in Features Defined in the Material Frame](#)

Heat Transfer Analysis with Moving Frames

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



- [Material and Spatial Frames](#)
 - [About Frames](#) in the *COMSOL Multiphysics Reference Manual*
-

In the heat transfer interfaces, most of the features (equations and variables) are defined in the spatial frame, excluding **Solid**, **Biological Tissue**, **Building Material**, **Shape Memory Alloy**, and **Battery Layers** features, which are defined in the material 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 postprocessing variables, and sometimes the 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
Battery Layers	Material
Bioheat	Material
Biological Tissue	Material
Building Material	Material
Cell Periodicity	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)

TABLE 3-8: DOMAIN PHYSICS NODES FOR FRAMES.

NODE NAME	DEFINITION FRAME
Shape Memory Alloy	Material
Solid	Material
Solid with Translational Motion	Material
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)
Boundary Pair	Material
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
Specular Mirror	Spatial
Surface-to-Ambient Radiation	Spatial
Symmetry	N/A
Temperature	Spatial
Thermal Contact	Material
Thermal Damage	Material

TABLE 3-9: BOUNDARY PHYSICS NODES FOR FRAMES.

NODE NAME	DEFINITION FRAME
Thermal Insulation	N/A
Thin Film	Spatial
Thin Layer	Material

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

DEFINITION FRAME OF EDGE AND POINT NODES

TABLE 3-10: 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 nonconstant 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](#), [Solid with Translational Motion](#), [Porous Medium](#), [Biological Tissue](#), [Building Material](#), [Shape Memory Alloy](#), [Battery Layers](#), [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.

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 checkbox is selected.

STREAMLINE DIFFUSION

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

CROSSWIND DIFFUSION

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

Inconsistent Stabilization

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

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



In the *COMSOL Multiphysics Reference Manual*:

- [Stabilization Techniques](#)
 - [Selection Information](#)
-

Heat Transfer and Fluid Flow Coupling

COMSOL Multiphysics offers physics interfaces for heat transfer and fluid flow computations. These interfaces have model inputs that make it possible to couple the physics. In addition, the **Nonisothermal Flow** multiphysics coupling node facilitates the coupling between heat transfer and fluid flow interfaces.

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

In this section:

- [Coupling Based on Model Inputs](#)
- [Adding Nonisothermal Flow Coupling in an Existing Model](#)
- [Nonisothermal Flow and Conjugate Heat Transfer Multiphysics Interfaces](#)

See [The Laminar Flow Interface](#), [The Turbulent Flow, Algebraic \$\gamma\$ Plus Interface](#), [The Turbulent Flow, L-VEL Interface](#), [The Turbulent Flow, k- \$\epsilon\$ Interface](#), [The Turbulent Flow, SST 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 bidirectional coupling between the two interfaces.

It is common to start a model with a single physics (for example, fluid flow), then implement the second one (for example, heat transfer). Then adding the **Nonisothermal Flow** multiphysics feature realizes the bidirectional coupling. It also redefines the consistent stabilization so that the multiphysics coupling effects are accounted for in the numerical stabilization.

With the Heat Transfer Module or the CFD Module, it accounts for the turbulence in the coupling. In particular, it modifies the effective thermal conductivity and implements thermal wall functions if the fluid flow model requires them. Those modifications affect the implementation of several heat transfer features. It allows to include work done by pressure changes and viscous dissipation, and Boussinesq approximation is supported. Finally some physics features are updated when the **Nonisothermal Flow** multiphysics feature is active. In particular, the **Interior Fan** and **Screen** fluid-flow features are updated to account for the multiphysics coupling.

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

Nonisothermal Flow and Conjugate Heat Transfer Multiphysics Interfaces

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

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

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

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

Boundary Wall Temperature

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

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

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

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

- `T`: general temperature variable that coincides with the wall temperature in most cases
- `TWall_u`: upside wall temperature defined by a **Wall** or an **Interior Wall** feature with turbulence only if a **Fluid** feature is defined on the upside of the wall.
- `TWall_d`: downside wall temperature defined by a **Wall** or an **Interior Wall** feature with turbulence only if a **Fluid** feature is defined on the downside of the wall.
- `Tu`: temperature on the upside of the boundary.
- `Td`: temperature on the downside of the boundary.
- `TExtFace`: external temperature of an external boundary defined by a thermally thick boundary condition.
- `TuWF`: temperature of the fluid in the turbulent boundary layer near the wall only if the **Fluid** feature is defined on the upside of the **Wall** feature.
- `TdWF`: temperature of the fluid in the turbulent boundary layer near the wall only if the **Fluid** feature is defined on the downside of the **Wall** feature.

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

- [Intersection Between a Wall Boundary Feature Selection and Interface Exterior Boundaries](#)
- [Interface Between a Fluid Domain Feature and a Solid Domain Feature Where a Wall Boundary Feature Is Active](#)
- [Interface Between two Fluid Domain Features Where an Interior Wall Boundary Feature Is Active](#)

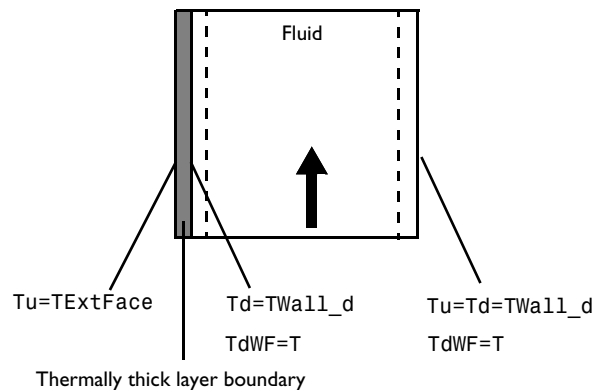
Depending on the turbulence model selected for the flow, wall functions are used or not:

- No turbulence model: no wall functions, and $Tu_{WF} = Tu$, $Td_{WF} = Td$.
- Turbulence models: wall functions detected by the Single Phase Flow 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

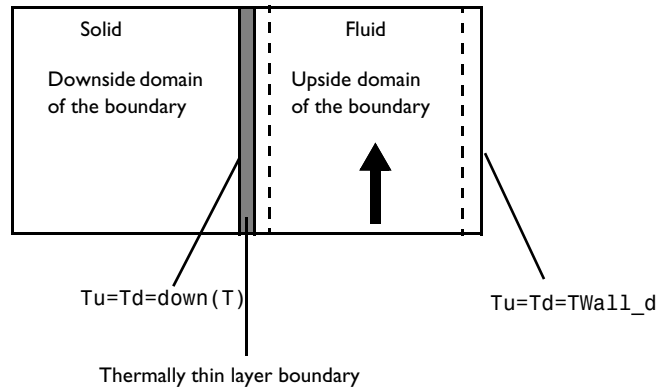
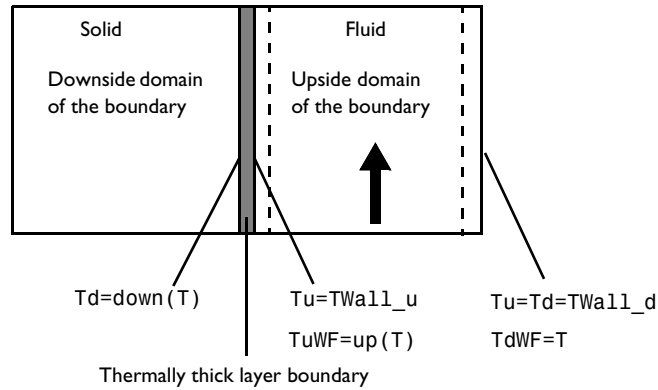
The following figure shows a configuration with a fluid domain, a thermally thick boundary condition on the left boundary, and a **Wall** feature on the right boundary. This example uses wall functions.



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

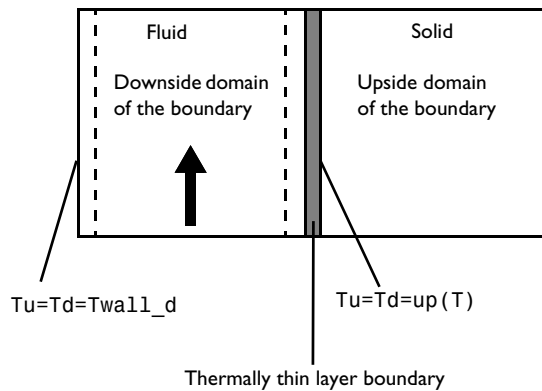
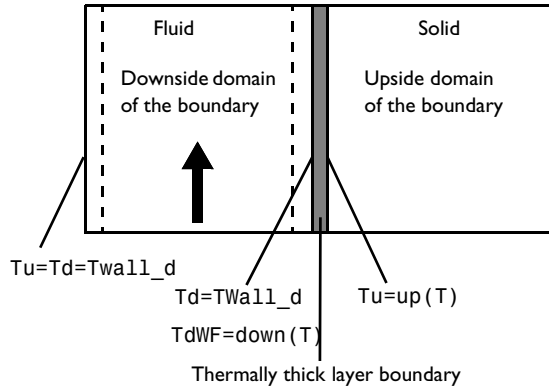
Solid in the Domain Downside the Wall

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.



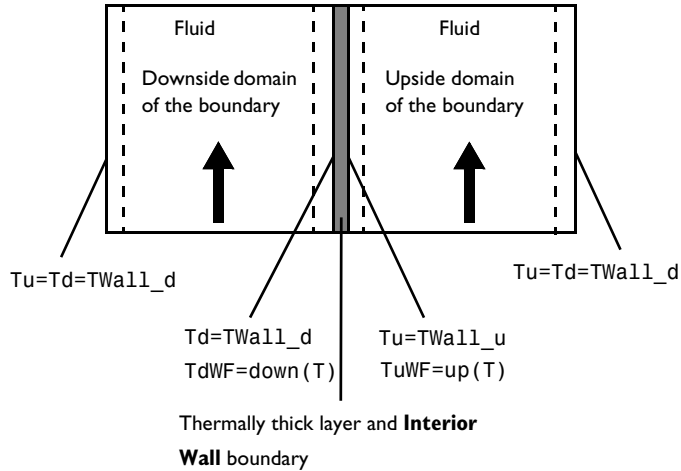
Solid in the Domain Upside the Wall

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 presmoothing and postsmoothing. 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 SCGS with Vanka option enabled, which is usually slower and uses more memory.

DEFAULT LINEAR SETTINGS FOR THE LUMPED THERMAL SYSTEM INTERFACE

When a **Lumped Thermal System** interface is coupled to one or many heat transfer interfaces using **Lumped Thermal Connector** nodes, the default linear solver is PARDISO if the total number of degree of freedom of the heat transfer and lumped thermal system interfaces is small. Otherwise a linear iterative GMRES solver with multiple preconditioner is used. The classical multigrid preconditioner from the heat transfer

interfaces is set for the heat transfer interface variables. And a direct preconditioner is used for the lumped thermal system variables.

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 512 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 α 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 **Once oer time step** 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 25 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

When a **Lumped Thermal System** interface is coupled to one or many heat transfer interface using **Lumped Thermal Connector** nodes, the lumped thermal system dependent variables and the temperature variables are gathered in the same segregated nodes if other degrees of freedom are solved for or in a **Fully Coupled** node.

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.

Time-Dependent Study Step

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, $\alpha_{phanecr}$, 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. First check the **Jacobian contribution** setting in the **Jacobian Contribution** section of the **Surface-to-surface Radiation** interface. The default value, **Only local contributions to radiosity**, results in a lumped Jacobian matrix and reduces a lot the required amount of memory. There are a few configurations where this formulation may lead to poor convergence. In this case, using the **Include contributions from total irradiation** option should fix the issue at the price of a much larger memory consumption use, in particular using the **Direct area integration** or **Hemicube** method. In this case:

- the Jacobian matrix of the discrete model is 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 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 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 assumes a weak coupling between temperature and radiosity variables solved by a Heat Transfer interface and a Surface-to-Surface Radiation interface, respectively. The default solver contains dedicated segregated groups for temperature and radiosity variables.

Heat and Moisture Transport

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

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

Moisture Flow

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

When it assumes a weak coupling between the flow and the moisture interfaces (typically no gravity force accounted for in the flow interface), the default solver contains dedicated segregated groups for moisture and flow dependent variables. Each uses the default linear solver of the corresponding interface, except that the 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 (`nx`, `ny`, `nz`), that always points from downside to upside. See [Tangent and Normal Variables](#) in the *COMSOL Multiphysics Reference Manual*. Note that the normal vector (`ht.nx`, `ht.ny`, `ht.nz`) may be oriented differently.
- `expr` is the quantity to be evaluated at the point `xd`. For example, it can be set to `T` to evaluate the temperature. There are others postprocessing variables defined on the extra dimension that can be found in the **Equation View** subnode of the node applied on the thin layer.

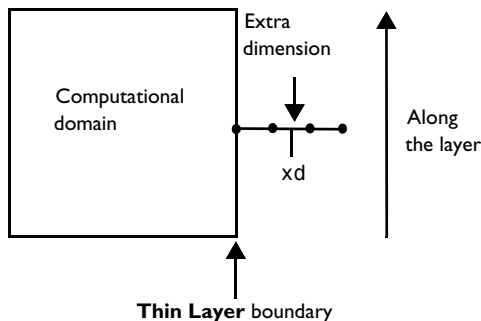


Figure 3-2: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results along the layer at the coordinate `xd`.

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

EVALUATION THROUGH THE LAYERED MATERIAL

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

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

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

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

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

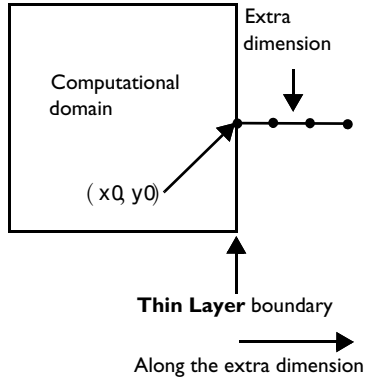


Figure 3-3: Schematic representation of a 2D geometry with a thin layer composed of three layers, with an evaluation of the results through the layer at the point (x_0, y_0) .

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 [nox](#)d in the *COMSOL Multiphysics Reference Manual* for details about the operators.

Evaluating the Bulk Temperature

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

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

- [Bulk Temperature Definition](#)
- [Evaluation of the Bulk Temperature in 3D Models](#)
- [Evaluation of the Bulk Temperature in 2D Models](#)

Bulk Temperature Definition

The bulk temperature T_b is defined as

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

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

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

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

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

Evaluation of the Bulk Temperature in 3D Models

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



See [ball, circle, filled cube, disk, and sphere](#) in the *COMSOL Multiphysics Reference Manual* for details about the operators.

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

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

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



See [General Extrusion](#) in the *COMSOL Multiphysics Reference Manual* for details about the operators.

Finally, the bulk temperature can be expressed as

$$T_b = \text{diskint}(R, \text{if}(\text{dom}==i, ut*T, 0), \text{tol}) / \text{diskint}(R, \text{if}(\text{dom}==i, ut, 0), \text{tol})$$

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

Evaluation of the Bulk Temperature in 2D Models

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

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



See [integrate](#) in the *COMSOL Multiphysics Reference Manual* for details about the operators.

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

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

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

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



See [at](#) in the *COMSOL Multiphysics Reference Manual* for details about the operators.

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



See [isinf](#) and [isnan](#) in the *COMSOL Multiphysics Reference Manual* for details about the operators.

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

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



See [General Extrusion](#) in the *COMSOL Multiphysics Reference Manual* for details about the operators.

Finally, the bulk temperature can be expressed as:

$$T_b = A/B$$

where

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

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

Ambient Properties

The ambient data available under the **Definitions** branch come from the processing of measured data from ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.), ASHRAE Weather Data Viewer 6.0 (©2017 ASHRAE, www.ashrae.org. Used with permission.), and Weather Data Viewer 2021 (©2021 ASHRAE, www.ashrae.org. Used with permission.) given as frequencies of observations, monthly and hourly averaged for several past years of observation. They provide the geographic position, and time-dependent weather conditions for about 8500 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, in 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)** or **Meteorological data (ASHRAE 2021)** 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. Three 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. The station details (World Meteorological Organization reference, location, and geographical coordinates) are displayed underneath.
- When **Around location** is selected, enter the **Latitude** and **Longitude** around which you want to find a station. Click on the **Set Weather Station** button to open the **Weather**

Station browser with the first 100 nearest stations, sorted by increasing distance from the specified location. The distance is calculated in kilometers using the haversine formula without consideration for the elevation. When the selection is performed, the station details (World Meteorological Organization reference, location, and geographical coordinates) are displayed underneath.

	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, From reference, or Around location, 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 https://ashrae-meteo.info/v2.0/.
	When a station is selected Around location, the Latitude and Longitude are decimal values, positive in the northern hemisphere and at the east of the Prime Meridian respectively. Both must be given without a unit to avoid double conversion. This is because the latitude and longitude values are 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

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 is 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** checkbox 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 checkbox 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 <i>COMSOL Multiphysics Reference Manual</i> 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**. Select the **Filter diurnal variation** checkbox to get temperature and relative humidity conditions without diurnal fluctuation. 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⁾	$T_{\text{amb}} = \langle T_{\text{station}} \rangle + \Delta T_{\text{wind}}$

TABLE 3-11: TEMPERATURE CONDITIONS.

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

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	$P_{0,\text{amb}} = \langle P_{0,\text{station}} \rangle - \sigma_{P_{\circ},\text{station}}$
High	$P_{0,\text{amb}} = \langle P_{0,\text{station}} \rangle + \sigma_{P_{\circ},\text{station}}$
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 can therefore exceed 1.



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 is 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.

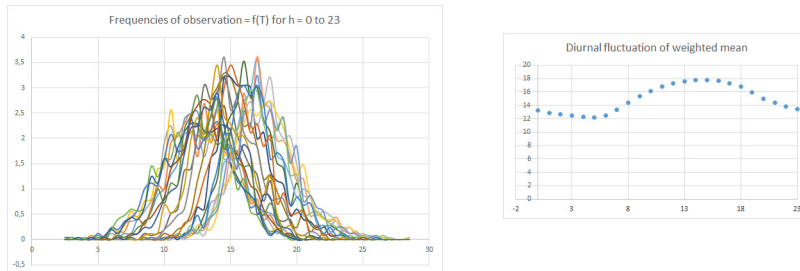


Figure 3-4: Computation of weighted mean from frequencies of observations for the diurnal fluctuations of temperature.

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 is 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 along with the diurnal fluctuation of the temperature

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



When the **Filter diurnal variation** checkbox is selected, the diurnal temperature fluctuation is smoothed out, and only the annual fluctuation remains. As relative humidity depends on temperature, it will display the same fluctuation.

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

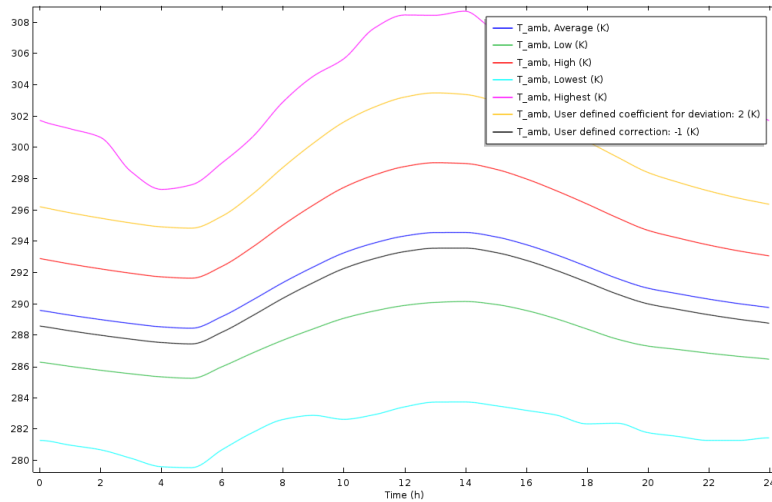


Figure 3-5: Comparison of ambient conditions for the temperature at New York/John F. Ke., on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.).

When diurnal temperature and relative humidity fluctuations have a low impact on the studied system, the **Filter diurnal variation** checkbox can be selected to smooth out the

diurnal fluctuations. This is illustrated in Figure 3-6 showing the variation of temperature over 1 month at New York/John F. Ke.,

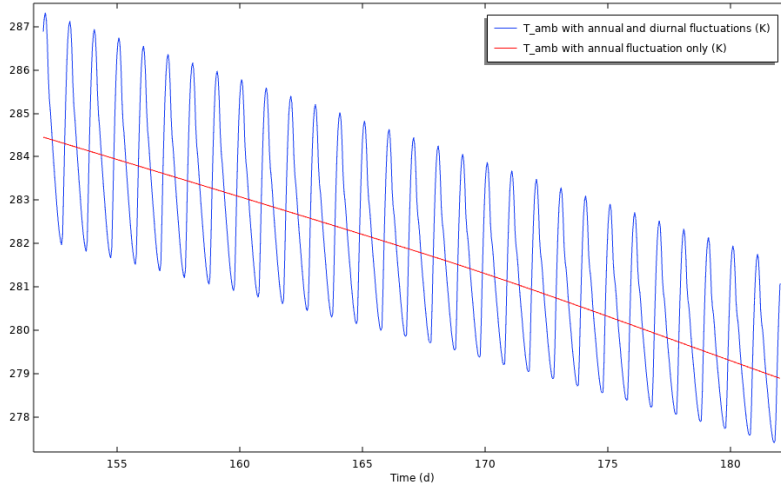


Figure 3-6: Comparison of ambient conditions for the temperature with and without diurnal fluctuation at New York/John F. Ke., for the month of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.).

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:

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

Here, 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 in Figure 3-7 for the variation of temperature over 1 day, at New York/John F. Ke., on the 1st of June.

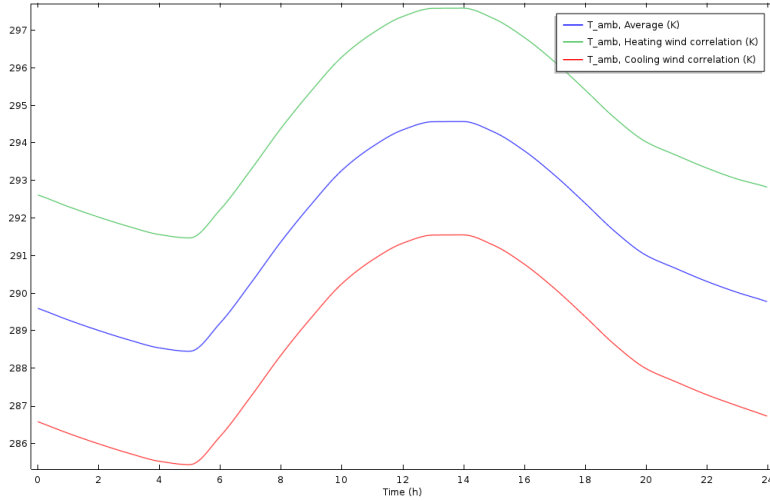


Figure 3-7: Comparison of heating and cooling wind correlations for the temperature at New York/John F. Ke., on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.).

CONDITIONS OF DEW-POINT TEMPERATURE

- Average:

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

- Low:

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

- High:

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

- Lowest:

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

- Highest:

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

where:

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

All these conditions are illustrated in Figure 3-8 for the variation of the dew-point temperature over 1 year at New York/John F. Ke.

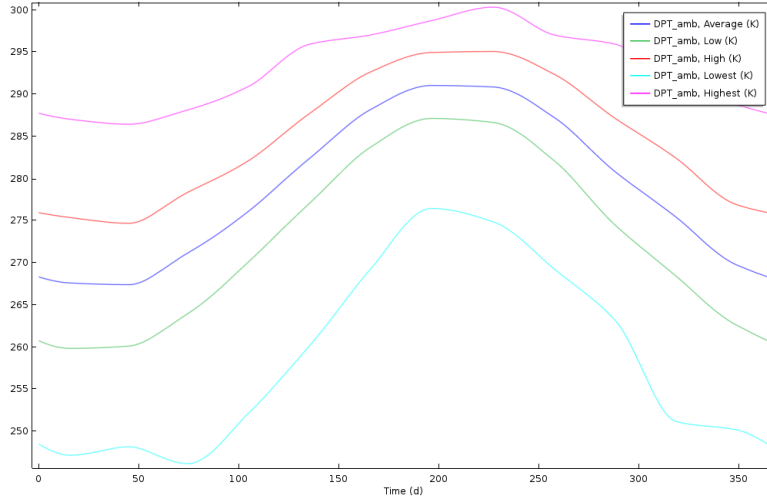


Figure 3-8: Comparison of the ambient conditions for the dew-point temperature at New York/John F. Ke., with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.).

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 in Figure 3-9 for the variation of the wind speed over 1 year at New York/John F. Ke.

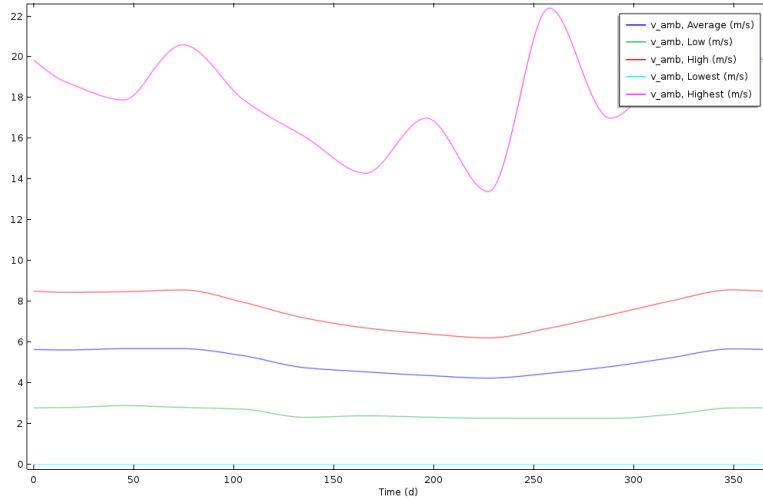


Figure 3-9: Comparison of the ambient conditions for the wind speed at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.).

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:

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

Here:

- $\langle P_{0, \text{station}} \rangle$ (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:

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

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 also be computed with the **Filter diurnal variation** checkbox cleared, as

illustrated in Figure 3-10 for New York/John F. Ke, on the 1st of June, for different ambient conditions.

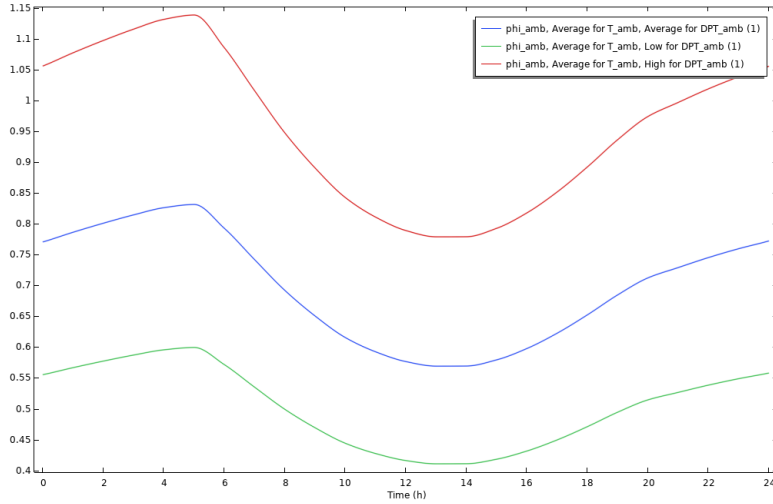


Figure 3-10: Diurnal fluctuations of relative humidity for different ambient conditions at New York/John F. Ke, on the 1st of June, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.).

Several conditions of relative humidity can be obtained from the selection of conditions for the temperature and dew-point temperature. The relative humidity can therefore exceed 1, as the red curve in Figure 3-10 shows.

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:

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

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 $I_{s, \text{amb}}$ (SI unit: W/m^2), defined by:

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

where $I_{\text{sn}, \text{station}}$ (SI unit: W/m^2) and $I_{\text{sh}, \text{station}}$ (SI unit: W/m^2) are respectively the observed values of the clear sky noon beam normal and horizontal diffuse solar irradiances.

Figure 3-11 illustrates the evolution of ambient solar irradiance for New York/John F. Ke, over the year.

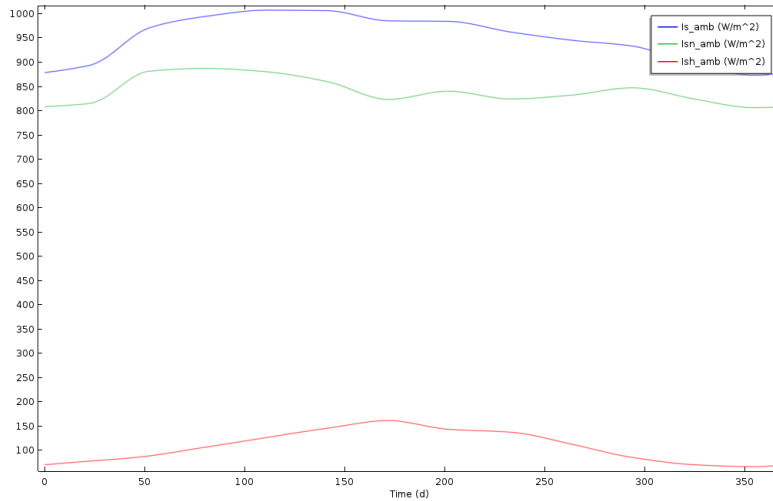


Figure 3-11: Decomposition of solar irradiance into normal and horizontal irradiance at New York/John F. Ke, with ASHRAE Weather Data Viewer 5.0 (©2013 ASHRAE, www.ashrae.org. Used with permission.).

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

Flux type:
Convective heat flux

Heat transfer coefficient:
External forced convection

Plate, averaged transfer coefficient

Plate length:
L 1 m

Velocity, fluid:
U Wind speed (ampr1)

Fluid:
Air

Absolute pressure:
 p_A Ambient absolute pressure (ampr1)

External temperature:
 T_{ext} Ambient temperature (ampr1)

Figure 3-12: 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**

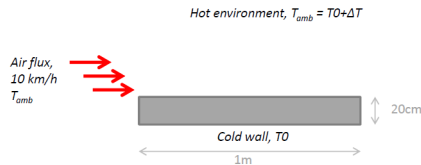
Modeling Heat Transfer by Radiation

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).

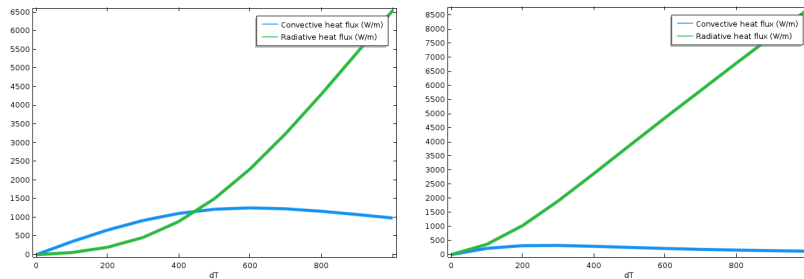


Figure 3-13: Integral of convective and radiative heat fluxes (W/m) along the top boundary, for two values of emissivity, as a function of the temperature difference.

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



See [Heat Transfer Variables](#) for the definition of the variables `q0` and `rflux` giving access to the convective and radiative heat fluxes on boundaries.

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

- Radiation in transparent media
- Radiation in participating media

Interfaces and Features for Radiation in Transparent Media

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

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

	SURFACE-TO-AMBIENT	SURFACE-TO-SURFACE
Description		
Physical and modeling assumptions		
Refractive index of domain	$n=1$	Any, uniform
Diffuse reflection	Yes	Yes
Specular reflection	No	With Ray shooting
Transmission	No	With Ray shooting
Shadowing effects in view factor computation	No	With Hemicube or Ray shooting
Wavelength dependence of surface properties	No	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 Specular Mirror 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)
⁽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		

Surface-to-Surface Radiation and Topology Consistency

When defining a surface-to-surface radiation model, for the boundaries that radiate only on one side, it is important to make sure each radiating surface receives radiation only on its radiating side. A boundary that radiates on one side should not receive radiation on its back. An optional topology check is enabled by default for the **Hemicube** and **Ray Shooting** methods. If it detects that the back side of a radiating boundary can be seen from a point computing the irradiation a warning message is displayed. This may be due to erroneous definition of the boundary radiation

directions or the opacity of the adjacent domains. The `rad.Fbackside` and `rad.Fbacksideu` variables can be plotted to verify the locations where the problem has been detected.

This may also be due to the combination of a relatively coarse mesh with the **High order mesh elements** option selected in the interface settings. Conversely, quadrilateral mesh elements combined with curved boundaries can create inconsistencies in view factor calculations when **High-order mesh** elements is not selected.

Note that checking the model consistency does not require additional computational time when the **Ray shooting** method is used. In order to speed up the computation, in particular with the **Hemicube** method, once the configuration has been verified, it is recommended to deselect **Check consistency** option.

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.

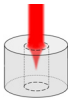
	ABSORBING MEDIA	ABSORBING-SCATTERING MEDIA	PARTICIPATING MEDIA
Description			
Physical and modeling assumptions			
Refractive index of domain	Any, uniform	Any, uniform	Any, uniform
Scattering in domain	N/A	Isotropic (PI method, DOM) Linear anisotropic (PI method, DOM) Anisotropic (DOM) Henyey–Greenstein (DOM)	Isotropic (Rosseland approximation, PI method, DOM) Linear anisotropic (PI method, DOM) Anisotropic (DOM) Henyey–Greenstein (DOM)

TABLE 3-16: MODELING OF HEAT TRANSFER BY RADIATION.

	ABSORBING MEDIA	ABSORBING-SCATTERING MEDIA	PARTICIPATING MEDIA
Wavelength dependency of emissivity	No	Yes (Solar and ambient / Multiple spectral bands) No (Constant)	Yes (Solar and ambient / Multiple spectral bands) No (Constant)
Dimension of the domain	3D, 2D axi, 2D	3D, 2D axi, 2D	3D, 2D axi, 2D
Example model	N/A	N/A	Glass Plate
License, interfaces and features			
License requirement	Heat Transfer Module	Heat Transfer Module	Heat Transfer Module
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	Opaque Surface Semitransparent Surface Incident Intensity Continuity of Interior Boundary (DOM) Periodic Condition Symmetry
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			

TABLE 3-16: MODELING OF HEAT TRANSFER BY RADIATION.

	ABSORBING MEDIA	ABSORBING-SCATTERING MEDIA	PARTICIPATING MEDIA
Numerical cost	Medium	Medium (PI method) High (DOM)	Low (Rosseland approximation) Medium (PI method) High (DOM)
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)

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.

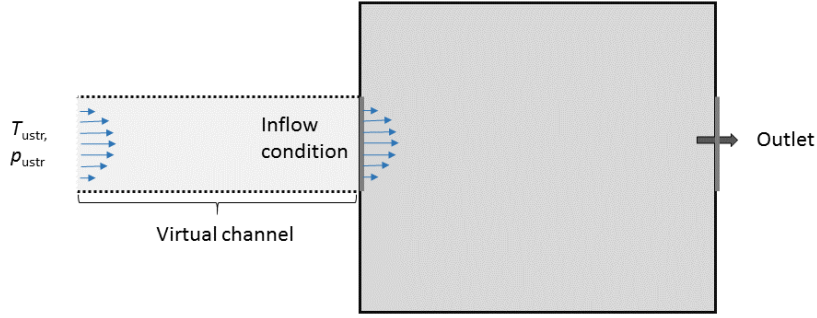


Figure 3-14: Representation of the virtual domain corresponding to an Inflow boundary condition.

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.



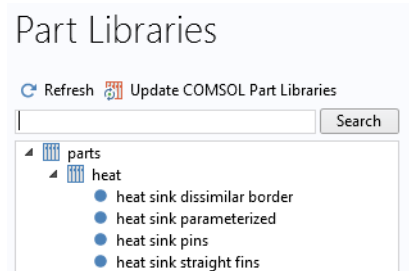
Part Libraries in the *COMSOL Multiphysics Reference Manual*

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 pins or straight fins, as shown in [Figure 3-15](#).

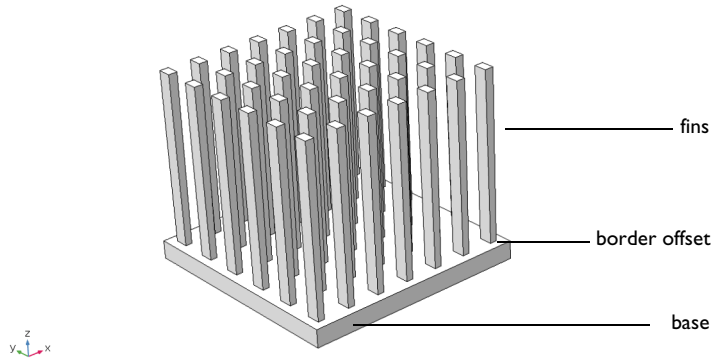
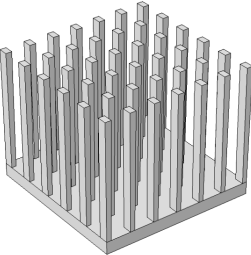
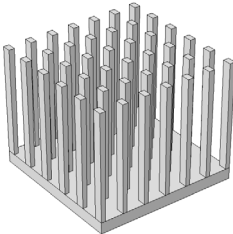
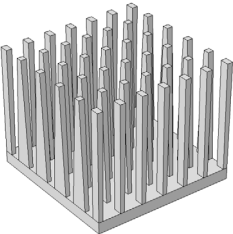
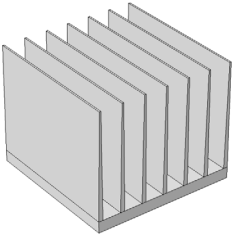


Figure 3-15: Heat sink components.

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 3-16](#). In addition, an offset from the base border can be added.

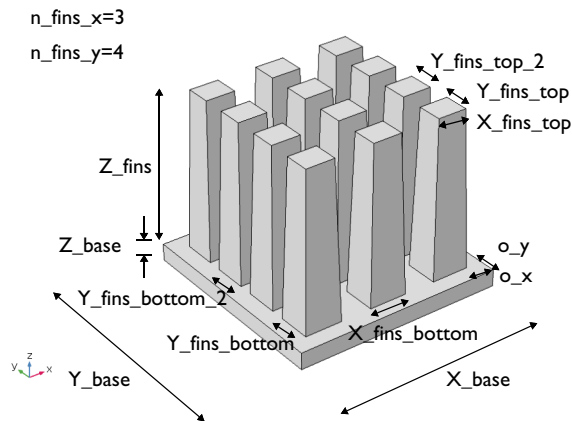


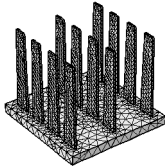
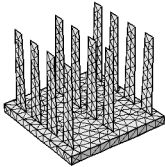
Figure 3-16: Fins and base parameters in Heat Sink — parameterized part.

3D AND SHELL FINS

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

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

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

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

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

STEP

A step can be defined in the x direction at the center on the base. The parameter `step_width` specifies the number of filled gaps from middle to border by the step, as shown on [Figure 3-17](#). This option is not available for shell fins (`shell=1`).

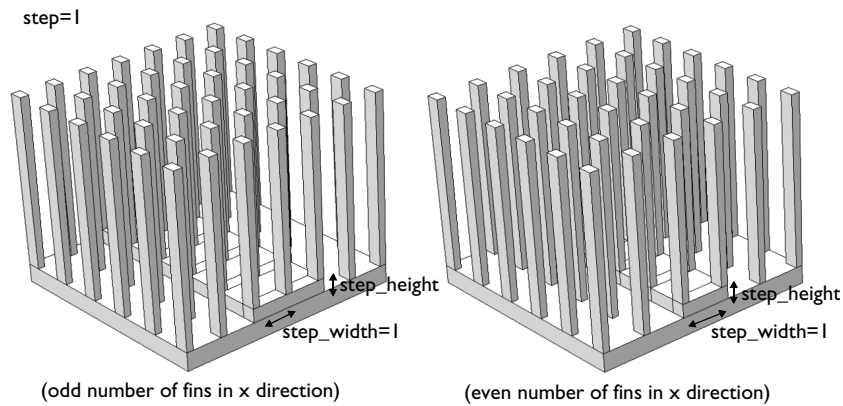


Figure 3-17: Step parameters in Heat Sink — parameterized part.

FILLET

Finally, a fillet transformation can be applied at the top and bottom of the fins, as shown on [Figure 3-18](#). The fillet transformation at the bottom of the fins is not available for shell fins (`shell=1`).

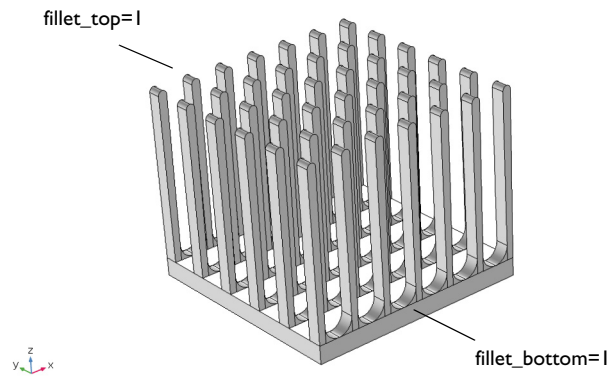


Figure 3-18: Fillet parameters in Heat Sink — parameterized part.

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 3-19](#).

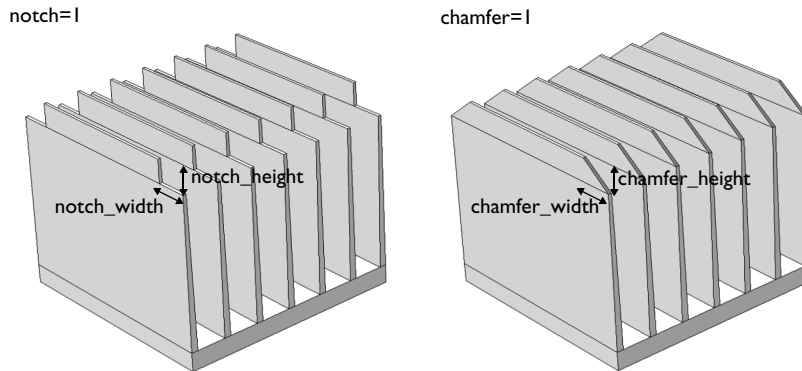


Figure 3-19: Notch and chamfer parameters in Heat Sink — straight fins part.

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 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 Cell Periodicity

- Theory for Thermal Connection
- Theory for Surface-to-Surface Radiation
- Theory for Orbital Thermal Loads
- Theory for Radiation in Participating Media
- Theory for Moisture Transport
- Theory for Moisture Transport in Porous Media
- Theory for the Heat Transfer Multiphysics Couplings
- Theory for Thermal Contact
- Out-of-Plane Heat Transfer
- Convective Heat Transfer Correlations
- Nucleate Pool Boiling Correlation
- Equivalent Thermal Conductivity Correlations
- Temperature Dependence of Surface Tension
- Heat Flux and Heat Balance
- Frames for the Heat Transfer Equations
- References

Foundations of the General Heat Transfer Equation

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

In this section:

- [Thermodynamic Description of Heat Transfer](#)
- [The Physical Mechanisms Underlying Heat Transfer](#)
- [The Heat Balance Equation](#)
- [Consistency with Mass and Momentum Conservation Laws](#)

Thermodynamic Description of Heat Transfer

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

EXTENSIVE PARAMETERS CHARACTERIZING A SYSTEM

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

- The *entropy*, S_{Ω} (SI unit: J),

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

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

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

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, $\boldsymbol{\tau}$, 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:

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

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:

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

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](#):

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

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: $1/\text{K}$)
- $\mathbf{S}$ 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:

$$Q_{\text{ted}} = -\alpha T : \frac{d\mathbf{S}}{dt} \quad (4-17)$$

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 an 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. 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. See [Saturation State](#) for the definition of saturation pressure p_{sat} as a function of temperature.

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. 36](#), the saturation pressure can be defined using the following expression:

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

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 the amount of dry air
- n_v is the 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.

See [Saturation State](#) for the definition of saturation pressure p_{sat} as a function of temperature.



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

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

where M_m represents the mixture molar mass and is defined by

$$M_m = X_a M_a + X_v M_v$$

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

Dynamic Viscosity

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

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

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

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

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. 39](#) for pure steam properties. The steam properties are based on the Industrial Formulation IAPWS-IF97.

The valid temperature range is $200 \text{ K} < T < 1200 \text{ K}$ for dry air properties and $273.15 \text{ K} < T < 873.15 \text{ K}$ for steam properties.

RESULTS AND ANALYSIS VARIABLES

These variables are provided to display the related quantities:

- Moisture content `xvap`.
- Vapor mass fraction `omega_moist`.
- Concentration of water vapor `c`.
- Relative humidity `phi`. This variable corresponds to the calculated ϕ with the system temperature and pressure.
- Saturation indicator `satInd`; this indicator is set to 1 if saturation has been detected ($\phi_w = 1$), and 0 if not.

FUNCTIONS

The following functions are defined and can be used as feature parameters as well as in postprocessing. Here, *feature* stands for `fluid` or `porous`, depending on whether the function is defined in the **Fluid** or in the **Porous Medium** feature:

- `ht.feature.fc(RH, T, pA)`, where RH is the relative humidity $0 \leq \phi_w \leq 1$, 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](#):

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



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

- `ht.feature.fxvap(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. 39](#), which provides steam properties based on the Industrial Formulation IAPWS-IF97. The temperature-dependency is as shown in [Figure 4-1](#).

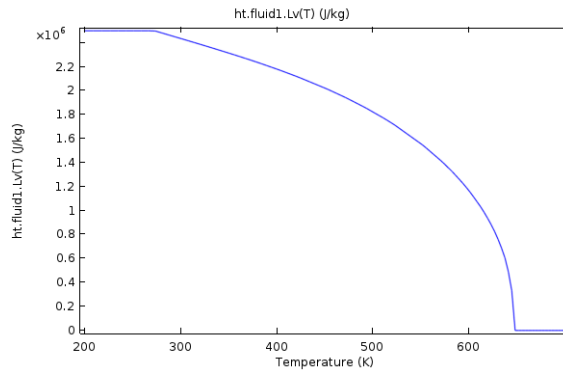


Figure 4-1: Latent heat of evaporation as a function of temperature.

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/m³)
- C_p is the specific heat capacity at constant pressure of the tissue (SI unit: J/(kg·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/m²)
- ρ_b is the blood density (SI unit: kg/m³)
- $C_{p,b}$ is the blood specific heat capacity at constant pressure (SI unit: J/(kg·K))
- ω_b is the blood perfusion rate (SI unit: 1/s)
- T_b is the arterial blood temperature (SI unit: K)
- Q_{met} is the metabolic heat source (SI unit: W/m³)

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 phenomena 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

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

for hyperthermia analysis, or by

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

for cryogenic analysis, with

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

evaluates the period of time when $T > T_{n,h}$ or the period of time when $T < T_{n,c}$. If $\alpha_{\text{necr}} > 0$, the tissue is necrotic because it already reached the necrosis temperatures $T_{n,h}$ or $T_{n,c}$ 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:

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

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

The fraction of necrotic tissue is then expressed by:

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

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

When considering heat transfer in a porous medium at the microscopic scale, two heat transfer equations can be established, for the solid and fluid phases. For undeformed immobile solids, [Equation 4-16](#) simplifies into:

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

and for a fluid domain where pressure work and viscous dissipation are neglected, [Equation 4-18](#) becomes:

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

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

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

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

When this hypothesis can be assumed, the heat transfer equation for porous media is derived from the mixture rule on energies appearing in solid and fluid heat transfer equations (see [Ref. 13](#)). This rule applies by multiplying the equation of the solid domain by the solid volume fraction, θ_s , multiplying the fluid equation by the porosity, ϵ_p , and summing resulting equations.

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

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

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

When Should Local Thermal Equilibrium and Nonequilibrium Be Considered?

The classical local equilibrium hypothesis in modeling heat transfer in porous media considers pointwise equality of solid and fluid temperatures as said in [Equation 4-40](#). The [Local Thermal Equilibrium](#) section below details the derivation of the energy equation considering such assumption that remains accurately sufficient for several applications. [Ref. 32](#) shows for instance that solid and fluid temperatures are equal in steady conduction problems where only prescribed temperature conditions are applied. Most slow motion problems can also assume equality of phase temperatures if volumetric internal heating do not differ in both materials.

In the case of conduction in porous plates, [Ref. 33](#) provides criteria based on the dimensionless Sparrow number, Sp , to indicate if temperature equilibrium is still valid or if a nonequilibrium point of view should be preferred. In [Ref. 34](#), the influence of the Darcy number, 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. 33](#) and [Ref. 34](#), 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 framework for heat transfer in porous media where evaluating the validity of the equilibrium hypothesis is not required anymore.

Local Thermal Equilibrium

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

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

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

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 $\mathbf{u}_f = \mathbf{u}/\epsilon_p$, where ϵ_p is the fluid's volume fraction, or equivalently the porosity.

- Q is the heat source (or sink). Add one or several heat sources as separate physics features.

For a steady-state problem the temperature does not change with time, and the terms with time derivatives of Equation 4-41 disappear.

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

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

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

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 :

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

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

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

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

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

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

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

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

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

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

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

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

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

Local Thermal Nonequilibrium

The [Local Thermal Nonequilibrium Interface](#) implements heat transfer in porous media for which the temperatures into the porous matrix and the fluid are not in equilibrium.

EQUATIONS FOR LOCAL NONEQUILIBRIUM HEAT TRANSFER

Nonequilibrium heat transfer in porous media for binary systems of rigid porous matrix and fluid phase are governed by a set of two equations. These are the usual heat equations for solids and fluids, multiplied by the volume fractions θ_s and $\epsilon_p = 1 - \theta_s$ respectively, and with an additional source term quantifying exchanged heat between both phases (2.12 and 2.13 in [Ref. 13](#)):

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

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

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

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

In these expressions:

- θ_s is the solid volume fraction (dimensionless)
- ε_p is the porosity (dimensionless)
- ρ_s and ρ_f are the solid and fluid densities (SI unit: kg/m³)
- $C_{p,s}$ and $C_{p,f}$ are the solid and fluid heat capacities at constant pressure (SI unit: J/(kg·K))
- $\mathbf{q}_s$ and $\mathbf{q}_f$ are the solid and fluid conductive heat fluxes (SI unit: W/m²)
- k_s and k_f are the solid and fluid thermal conductivities (SI unit: W/(m·K))
- q_{sf} is the interstitial convective heat transfer coefficient (SI unit: W/(m³·K))
- Q_s and Q_f are the solid and fluid heat sources (SI unit: W/m³)
- $\mathbf{u}$ is the fluid velocity vector (SI unit: m/s). It should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area. The average linear velocity (the velocity within the pores) can be calculated as $\mathbf{u}_f = \mathbf{u}/\varepsilon_p$, where ε_p is the fluid's volume fraction, or equivalently the porosity.

The [Porous Medium](#) feature with **Porous medium type** set to **Local thermal equilibrium** adds the exchanged opposite heat sources $q_{sf}(T_f - T_s)$ and $q_{sf}(T_s - T_f)$ that one phase receives from or releases to the other when respective temperatures differ. The porous temperature, T , has the following definition ([Ref. 35](#)):

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

CORRELATION FOR THE INTERSTITIAL CONVECTIVE HEAT TRANSFER COEFFICIENT

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

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

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

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

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

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

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

$$Nu = 2.0 + 1.1 Pr^{1/3} Re_p^{0.6}$$

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

$$Pr = \frac{\mu C_{p,f}}{k_f} \quad Re_p = \frac{d_{pe} \rho_f \|\mathbf{u}\|}{\mu}$$

VOLUMETRIC AND SURFACE THERMAL CONDITIONS

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

Packed Bed of Pellets

For the modeling of packed beds of pellets, [The Heat Transfer in Packed Beds Interface](#) interface implements a hybrid multiscale model of local thermal nonequilibrium.

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

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

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

EQUATIONS FOR HEAT TRANSFER IN A PACKED BED OF PELLETS

Macroscopic Heat Transfer in the Fluid

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

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

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

In these expressions:

- ε_p is the packed bed porosity (dimensionless)
- ρ_f is the fluid density (SI unit: kg/m^3)
- $C_{p,f}$ is the fluid heat capacity at constant pressure (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$)
- $\mathbf{q}_f$ is the fluid conductive heat flux (SI unit: W/m^2)
- k_f is the fluid thermal conductivity (SI unit: $\text{W}/(\text{m}\cdot\text{K})$)
- $Q_{pe,f}$ is the pellet-fluid heat exchange term (SI unit: $\text{W}/(\text{m}^3\cdot\text{K})$)
- Q_f is any fluid heat source (SI unit: W/m^3)
- $\mathbf{u}$ is the fluid velocity vector (SI unit: m/s). It should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area. The average linear velocity (the velocity within the pores) can be calculated as $\mathbf{u}_f = \mathbf{u}/\varepsilon_p$, where ε_p is the fluid's volume fraction, or equivalently the porosity.

Microscopic Heat Transfer in the Pellets

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

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

where

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

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

- ρ_{pe} is the density of the porous matrix within the pellet (SI unit: kg/m³)
- $C_{p, pe}$ is the volumetric heat capacity at constant pressure of the porous matrix within the pellet (SI unit: J/(kg·K))
- ε_{pe} is the porosity within the pellet (dimensionless)
- T_{pe} is the temperature along the radial coordinate of the pellet (SI unit: K)
- r_{pe} is outer radius of the pellet (SI unit: m)
- $k_{pe, eff}$ is the effective thermal conductivity of the pellet filled with fluid (SI unit: W/(m·K)), defined by

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

- k_{pe} is the thermal conductivity of the porous matrix within the pellet (SI unit: W/(m·K))
- Q_{pe} is any heat source applied to the pellet (SI unit: W/m³)

Coupling Condition at Pellet-Fluid Interface

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

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

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

with $h_{pe, f}$ the interstitial heat transfer coefficient (SI unit: W/(m²·K)), defined as in the standard local thermal nonequilibrium model.

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

$$Q_{pe, f} = -S_b q_{pe|r=1}$$

with S_b the specific surface area of the pellets. For a packed bed of spherical pellets, it is defined as

$$S_b = \frac{6(1 - \varepsilon_p)}{d_{pe}}$$

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

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

When continuity of temperatures at the fluid-pellet interface can be assumed, that is, when

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

then the interstitial heat flux can be rewritten as

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

Average Temperature in the Packed Bed

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

$$T = \frac{(1 - \varepsilon_p) \frac{\int_V (\rho C_p)_{\text{pe, eff}} T_{\text{pe}} dV}{V_{\text{pe}}} + \varepsilon_p \rho_f C_{p, f} T_f}{(1 - \varepsilon_p) \frac{\int_V (\rho C_p)_{\text{pe, eff}} dV}{V_{\text{pe}}} + \varepsilon_p \rho_f C_{p, f}}$$

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:

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

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

The different quantities appearing here are:

- ρ_g and ρ_l (SI unit: kg/m^3) the moist air and liquid water densities.
- $C_{p, g}$ 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:

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

- where s_l 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.
- $\mathbf{Q}$ (SI unit: W/m^3), defined as the sum of the diffusive flux of enthalpy and of the liquid capillary flux:

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

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 immiscible so that each phase is located in a distinct domain. The simulation predicts the position of the phase change interface and the associated thermal effects. The following sections present the two numerical methods available to do the modeling:

- [Arbitrary Lagrangian-Eulerian Method](#)
- [Apparent Heat Capacity Method](#)

Arbitrary Lagrangian-Eulerian Method

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

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

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

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

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

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

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

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.

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

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

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

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

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

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

Apparent Heat Capacity Method

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

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

The [Phase Change Material](#) domain condition should be used on both phases domains, to solve the heat equation after specifying the properties of the phase change material according to the *apparent heat capacity* formulation.

Instead of adding a latent heat L in the energy balance equation exactly when the material reaches its phase change temperature T_{pc} , it is assumed that the transformation occurs in a temperature interval ΔT . Within this interval, the material is transformed from phase 1 to phase 2. The corresponding volume fractions of the phases are represented by the phase transition function $\alpha_{1 \rightarrow 2}$, where $\theta_1 = 1 - \alpha_{1 \rightarrow 2}$ is the volume fraction of the material before transition and $\theta_2 = \alpha_{1 \rightarrow 2}$ the volume fraction of the material after transition. See the section [Phase Transition Function](#) for details about the form of $\alpha_{1 \rightarrow 2}$.

The density, ρ , and the specific enthalpy, H , are expressed by:

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

$$H = \frac{1}{\rho} (\theta_1 \rho_1 H_1 + \theta_2 \rho_2 H_2)$$

Differentiating with respect to temperature, this equality provides the following formula for the specific heat capacity:

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

which becomes, after some formal transformations:

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

The mass fraction, α_m , is defined from ρ_1 , ρ_2 and θ_1 , θ_2 according to:

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

The specific heat capacity is the sum of an equivalent heat capacity C_{eq} :

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

and the distribution of latent heat C_L :

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



In the ideal case, when $\alpha_{1 \rightarrow 2}$ 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

$$C_L(T) = L \frac{d\alpha_m}{dT} \quad (4-49)$$

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:

$$H = \theta_1 H_1 + \theta_2 H_2$$

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:

$$\alpha_m = \frac{1}{2} \frac{\theta_2 - \theta_1}{\theta_1 + \theta_2}$$

PHASE TRANSITION FUNCTION

How the phase transition from phase 1 to phase 2 takes place is described by the phase transition function $\alpha_{1 \rightarrow 2}$. By default, COMSOL uses a smoothed step function with a continuous second derivative (Heaviside function). It describes a continuous transition from phase 1 to phase 2 within the transition interval ΔT around the phase transition temperature T_{pc} . Besides the frequently used Heaviside function, a linear function is also common and available as predefined option.

Figure 4-2 illustrates the functions and the corresponding parameters.

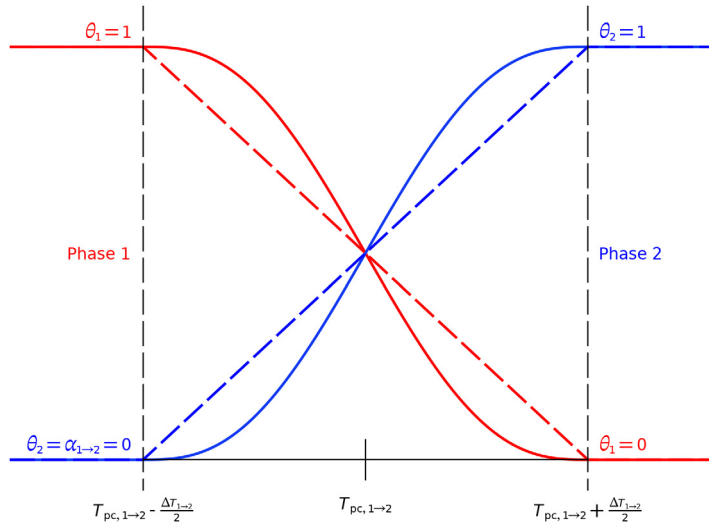


Figure 4-2: Phase volume fractions using Heaviside (solid) and linear (dotted) phase transition function.

With a user-defined phase transition function, the volume fraction of phase 2 is explicitly specified. For reasonable results, the value range of $\alpha_{1 \rightarrow 2}$ should be between 0 and 1 satisfying $\theta_1 + \theta_2 = 1$ and the function should be differentiable to correctly approximate the latent heat contribution, which is calculated from the derivative of this function by Equation 4-49. A user-defined function can also be used to describe a phase transition where not the complete volume fraction of one phase changes into the other, that is, there is a residual phase.

Theory for Heat Transfer in Building Materials

The [Heat Transfer in Building Materials Interface](#) solves for the following equations derived from [Ref. 16](#):

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = Q \quad (4-50)$$

$$\mathbf{q} = -(k_{\text{eff}} \nabla T + L_v \delta_p \nabla(\phi_w p_{\text{sat}})) \quad (4-51)$$

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:

$$k_{\text{eff}} = k_s \left(1 + \frac{bw(\phi_w)}{\rho_s} \right)$$

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. See [Saturation State](#) for the definition of saturation pressure p_{sat} as a function of temperature.

Theory for Harmonic Heat Transfer

When submitted to periodic sinusoidal heat loads at a given frequency, the temperature response of a body can, in some cases, be assumed periodic, sinusoidal, and of same frequency around an equilibrium temperature. If the temperature changes are small enough or if the thermal properties are constant, this time-dependent periodic problem may be replaced by an equivalent linear steady-state problem in the frequency domain, which is far less computationally expensive. The Heat Transfer interfaces support such frequency domain studies via the **Thermal Perturbation, Frequency Domain** study sequence.

In complex notations, the aforementioned temperature decomposition is expressed as:

$$T(t) = T_0 + T^* e^{j\omega t} \quad (4-52)$$

where T_0 is the equilibrium temperature that verifies the steady-state heat transfer equation and may come from the solution of a previous study, T^* is the complex amplitude of the harmonic perturbation around T_0 , and ω is the angular frequency, related to the ordinary frequency, f , according to

$$\omega = 2\pi f$$

Note: The amplitude, T^* , is complex valued since it includes the phase term $e^{j\varphi}$.

From the temperature decomposition in [Equation 4-52](#), and according to the heat transfer equation in [Equation 4-16](#), heat transfer by conduction in solids is then governed by:

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

where $Q' e^{j\omega t}$ is the harmonic perturbation in domain around an average heat source, Q . Removing the terms of the steady-state heat transfer equation satisfied by T_0 , and simplifying by $e^{j\omega t}$, this reduces to:

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

which is the governing equation for T^* . For constant material properties, the linearized form reads:

$$j\omega\rho_0 C_{p,0} T^* + \nabla \cdot (-k_0 \nabla T^*) = Q^* \quad (4-53)$$

and for nonlinear material properties:

$$j\omega\rho_0 C_{p,0} T^* + \nabla \cdot \left(-k_0 \nabla T^* - T^* \left(\frac{\partial k}{\partial T} \right)_{T_0} \nabla T_0 \right) = Q^* \quad (4-54)$$

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 4-53](#) or [Equation 4-54](#), 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:

$$m \overline{C_p} \frac{dT}{dt} + \int_S (\mathbf{n} \cdot \mathbf{q}) ds = \int_V Q dv \quad (4-55)$$

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.

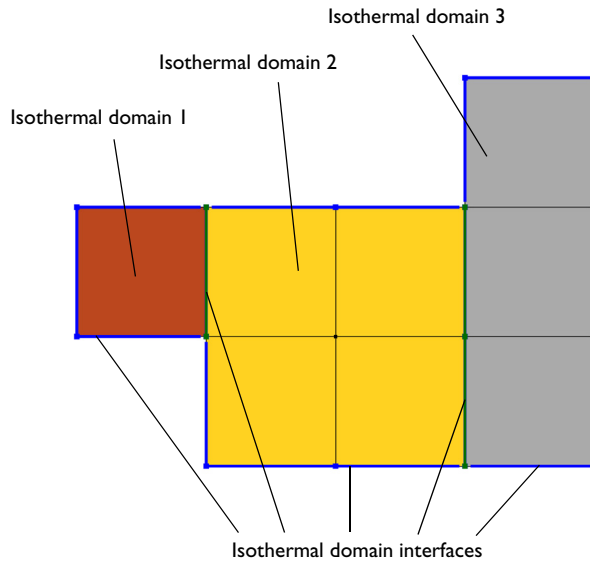


Figure 4-3: Interfaces at exterior boundaries of each Isothermal Domain.

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

$$-\mathbf{n}_d \cdot \mathbf{q}_d = \phi_{d \rightarrow u} H_d - \phi_{u \rightarrow d} H_u \quad (4-56)$$

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

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -h(T_u - T_d) \quad (4-57)$$

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

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -\frac{T_u - T_d}{R_{t,s}} \quad (4-58)$$

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.

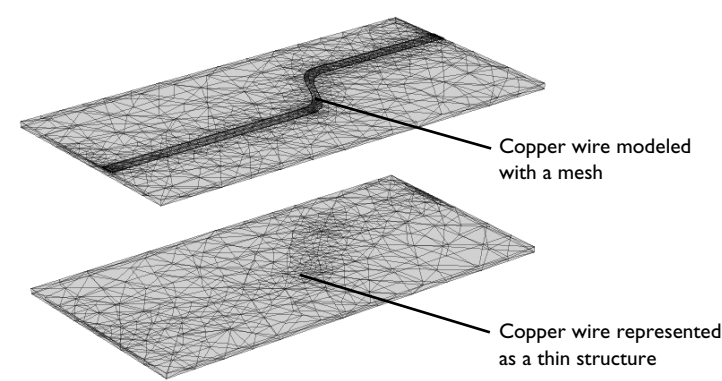


Figure 4-4: Modeling a copper wire as a domain (top) requires a denser mesh compared to modeling it as a boundary with a conductive layer (bottom).

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:

$$\nabla T = \nabla_{\mathbf{t}}T + \nabla_{\mathbf{n}}T \quad (4-59)$$

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 4-59 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

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

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

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

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

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

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

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

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

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:

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

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 4-59](#), the energy equation becomes

$$\rho \frac{\partial E_s}{\partial t} + \rho \mathbf{u} \cdot (\nabla_{\mathbf{t}} E_s + \nabla_{\mathbf{n}} E_s) + \nabla \cdot (\mathbf{q} + \mathbf{q}_r) = -(\sigma : \mathbf{D}) + Q \quad (4-60)$$

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 4-60](#) comes along with Fourier's law of conduction

$$\mathbf{q} = -k(\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \quad (4-61)$$

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

$$T_d = (T_s)_{L=0} \quad T_u = (T_s)_{L=d_s} \quad (4-62)$$

Here, d_s is the length of the extra dimension, or equivalently the thickness of the thin structure, and T_u and T_d are the temperature at the upside and the downside of the thin structure.

THERMALLY THIN APPROXIMATION

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

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

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

With these assumptions, Equation 4-15 becomes:

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

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

where d_s is the layer thickness (SI unit: m). The heat source Q is a heat source distributed in the layers 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:

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

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

THERMALLY THICK APPROXIMATION

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

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

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

With these assumptions, [Equation 4-15](#) becomes:

$$d_s \rho \frac{\partial E}{\partial t} + d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{n}} E + \nabla_{\mathbf{n}} \cdot (\mathbf{q}_s + \mathbf{q}_r) = -d_s (\sigma : \mathbf{D}) + d_s Q + q_0 \quad (4-66)$$

$$\mathbf{q}_s = -d_s k \nabla_{\mathbf{n}} T \quad (4-67)$$

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 4-66](#) and [Equation 4-67](#) have an additional factor, d_z , to account for the out-of-plane thickness.

When [Equation 4-66](#) 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:

$$-\mathbf{n} \cdot \mathbf{q} = d_s Q_s - d_s \rho \frac{\partial E}{\partial t} - (d_s \rho \mathbf{u} \cdot \nabla_{\mathbf{n}} E) - \nabla_{\mathbf{n}} \cdot (\mathbf{q}_s + \mathbf{q}_r) \quad (4-68)$$

To evaluate the normal gradient operation, $\nabla_{\mathbf{n}}$, the 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 as

$$\rho C_p \frac{\partial T}{\partial t} \approx \rho C_p \frac{\partial T_{1/2}}{\partial t} = \frac{1}{2} \rho C_p \frac{\partial T_u}{\partial t} + \frac{1}{2} \rho C_p \frac{\partial T_d}{\partial t}$$

leading to

$$-\mathbf{n}_d \cdot \mathbf{q}_d = \frac{d_s Q}{2} - \rho C_p \left(\frac{d_s}{2} \frac{\partial T_d}{\partial t} + \mathbf{u}_d \cdot \mathbf{n}_d T_d \right) - \left(-k_s \frac{T_u - T_d}{d_s} \right) - (\mathbf{q}_{r,d} \cdot \mathbf{n}_d) \quad (4-69)$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = \frac{d_s Q}{2} - \rho C_p \left(\frac{d_s}{2} \frac{\partial T_u}{\partial t} + \mathbf{u}_u \cdot \mathbf{n}_u T_u \right) - \left(-k_s \frac{T_d - T_u}{d_s} \right) - (\mathbf{q}_{r,u} \cdot \mathbf{n}_u) \quad (4-70)$$

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:

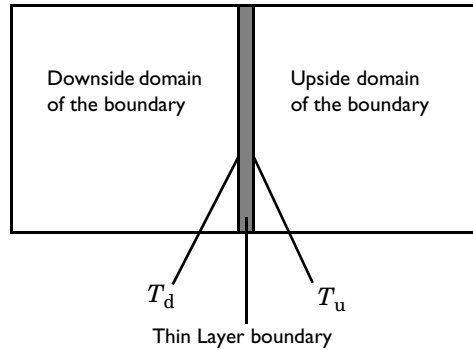


Figure 4-5: Upside and downside temperatures at a thin layer applied on an interior boundary. The thin layer is represented by the gray domain.

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:

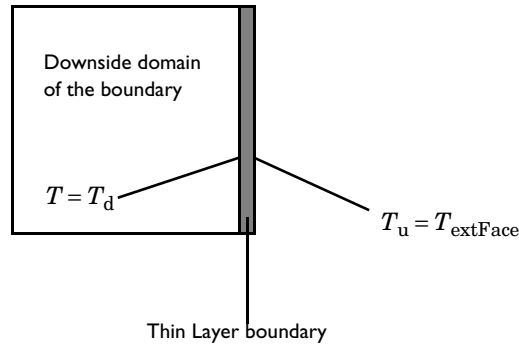


Figure 4-6: Upside and downside temperatures at a thin layer applied on an exterior boundary.

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 4-60 to give:

$$\rho_{si} C_{p, si} \frac{\partial T_s}{\partial t} + \nabla_{\mathbf{t}} \cdot \mathbf{q}_{si} = Q_{si} \quad (4-71)$$

$$\mathbf{q}_{si} = -k_{si} (\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \quad (4-72)$$

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

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

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 4-71](#) and [Equation 4-72](#).

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 4-63](#) to [Equation 4-65](#). Inside the thin layer, the heat equation becomes:

$$d_s \rho C_{p,s} \frac{\partial T}{\partial t} + \nabla_{\mathbf{t}} \cdot \mathbf{q}_s = d_s Q_s + q_0 \quad (4-73)$$

$$\mathbf{q}_s = -d_s k \nabla_{\mathbf{t}} T \quad (4-74)$$

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 4-73](#) and [Equation 4-74](#) 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 4-65](#), is received from the layer:

$$-\mathbf{n} \cdot \mathbf{q} = d_s Q_s - d_s \rho C_{p,s} \frac{\partial T}{\partial t} - \nabla_{\mathbf{t}} \cdot \mathbf{q}_s \quad (4-75)$$

This formulation may be used to model efficiently a shell with multiple sandwiched layers with different material properties and thicknesses, by approximating the homogenized material properties as weighted averages of constant properties within each layer:

- The homogenized density is defined as

$$\rho = \frac{\sum_i \rho_i d_{s,i}}{\sum_i d_{s,i}}$$

- The homogenized heat capacity is defined as

$$C_p = \frac{\sum_i C_{p,i} d_{s,i}}{\sum_i d_{s,i}}$$

- The homogenized thermal conductivity is defined as

$$k = \frac{\sum_i (\mathbf{rot}_i \cdot \mathbf{A})^T \cdot k_i \cdot (\mathbf{rot}_i \cdot \mathbf{A}) d_{s,i}}{\sum_i d_{s,i}}$$

with $\mathbf{rot}_i$ the rotation tensor of layer i :

$$\mathbf{rot}_i = \begin{bmatrix} \cos(\theta) & \sin(\theta) & 0 \\ -\sin(\theta) & -\cos(\theta) & 0 \\ 0 & 0 & 1 \end{bmatrix}$$

and $\mathbf{A}$ the transformation matrix between the local and global coordinates systems:

$$\mathbf{A} = \begin{bmatrix} t_{1,x} & t_{2,x} & n_x \\ t_{1,y} & t_{2,y} & n_y \\ t_{1,z} & t_{2,z} & n_z \end{bmatrix}$$



See [Thin Layer \(Heat Transfer Interface\)](#) and [Solid \(Heat Transfer in Shells Interface\)](#) with **Layer type** set as **Thermally thin approximation** for more information about the boundary feature solving [Equation 4-75](#). See [The Heat Transfer in Shells Interface](#) for more information about the physics interface solving [Equation 4-73](#).



- *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 4-68](#) and gives

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -\frac{1}{2}d_s \rho_s C_{p,s} \frac{\partial T_d}{\partial t} - k_s \frac{T_u - T_d}{d_s} + \frac{1}{2}d_s Q_s \quad (4-76)$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = -\frac{1}{2}d_s \rho_s C_{p,s} \frac{\partial T_u}{\partial t} - k_s \frac{T_d - T_u}{d_s} + \frac{1}{2}d_s Q_s \quad (4-77)$$

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 4-78](#) and [Equation 4-79](#):

$$d_{\text{tot}} = \sum_{j=1}^n d_{sj} \quad (4-78)$$

$$k_{\text{tot}} = \frac{d_{\text{tot}}}{\sum_{j=1}^n \frac{d_{sj}}{k_{sj}}} \quad (4-79)$$

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 4-76](#) and [Equation 4-77](#).

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 4-60](#) to give:

$$\rho C_p \frac{\partial T_s}{\partial t} + \rho C_p \mathbf{u} \cdot (\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) + \nabla_{\mathbf{t}} \cdot \mathbf{q}_f = Q_f \quad (4-80)$$

$$\mathbf{q}_f = -k(\nabla_{\mathbf{t}} T_s + \nabla_{\mathbf{n}} T_s) \quad (4-81)$$

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

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

where T_s is an auxiliary dependent variable defined on the product space. The remaining quantities are recalled below:

- ρ is the density (SI unit: kg/m^3)

- C_p is the heat capacity (SI unit: J/(kg·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 4-80](#) and [Equation 4-81](#).

THERMALLY THIN APPROXIMATION

The thermally thin approximation is derived from [Equation 4-63](#) to [Equation 4-65](#). Inside the thin layer, the heat equation becomes:

$$d_f \rho C_p \left(\frac{\partial T}{\partial t} + \mathbf{u} \cdot \nabla_{\mathbf{t}} T \right) + \nabla_{\mathbf{t}} \cdot \mathbf{q}_f = d_f Q_f + q_0 \quad (4-82)$$

$$\mathbf{q}_f = -d_f k \nabla_{\mathbf{t}} T \quad (4-83)$$

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 4-73](#) and [Equation 4-74](#) 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 4-65](#), is received from the layer:

$$-\mathbf{n} \cdot \mathbf{q} = d_f Q_f - d_f \rho C_p \frac{\partial T}{\partial t} - d_f \rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T - \nabla_{\mathbf{t}} \cdot \mathbf{q}_f \quad (4-84)$$



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 4-82](#). See [The Heat Transfer in Films Interface](#) for more information about the physics interface solving [Equation 4-84](#).

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: m²), 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 4-63](#) to [Equation 4-65](#) 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:

$$d_{\text{fr}}(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + d_{\text{fr}} \rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T + \nabla_{\mathbf{t}} \cdot \mathbf{q}_{\text{fr}} = d_{\text{fr}} Q + q_0 \quad (4-85)$$

$$\mathbf{q}_{\text{fr}} = -d_{\text{fr}} k_{\text{eff}} \nabla_{\mathbf{t}} T \quad (4-86)$$

Here $(\rho C_p)_{\text{eff}}$ is the effective heat capacity at constant pressure of the fracture-fluid volume, ρ is the fluid's density, C_p is the fluid's heat capacity at constant pressure, $\mathbf{q}_{\text{fr}}$ is the conductive heat flux in the fracture-fluid volume, k_{eff} is the effective thermal conductivity of the fluid-fracture mixture, and Q is a possible heat source.

From the point of view of the domain, the following heat source, derived from [Equation 4-65](#), is received from the fracture:

$$-\mathbf{n} \cdot \mathbf{q} = d_{\text{fr}} Q_0 - d_{\text{fr}} (\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} - d_{\text{fr}} \rho C_p \mathbf{u} \cdot \nabla_{\mathbf{t}} T - \nabla_{\mathbf{t}} \cdot \mathbf{q}_{\text{fr}} \quad (4-87)$$



See [Fracture \(Heat Transfer Interface\)](#) and [Porous Medium \(Heat Transfer in Shells Interface\)](#) for more information about the boundary feature solving [Equation 4-87](#). See [The Heat Transfer in Fractures Interface](#) for more information about the physics interface solving [Equation 4-85](#).

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:

$$\int_{S(R)} Q ds = A_1 Q_1 - A_1 \rho_1 C_{p,1} + \frac{\partial T}{\partial t} - \nabla_{\mathbf{t}} \cdot \mathbf{q}_1 \quad (4-88)$$

$$\mathbf{q}_1 = -A_1 k_1 \nabla_{\mathbf{t}} T \quad (4-89)$$

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	$-\frac{\Delta T}{R_{\text{conv}}}$	$-P$	P
Radiative Thermal Resistor	$-\frac{\Delta T}{R_{\text{rad}}}$	$-P$	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	$P_{1,\text{src}} + P_{2,\text{src}}$	$P_{1,\text{src}}$	$P_{2,\text{src}}$

Note that for thermal resistors components, the heat rate at port p1 has the same sign as the temperature difference ΔT across the component. In practice, when the temperature at port p2 is higher than the temperature at port p1 ($\Delta T > 0$), applying conductive heat transfer between these two ports is equivalent to apply a heat source (positive heat rate P_{p1}) at port p1.

For the heat pipe component, the port p1 should correspond to the evaporator side (hot side), and the port p2 should correspond to the condenser side (cold side). Therefore, the temperature difference ΔT across the component should be negative in

normal operating conditions. With this setting, P_{p1} is always negative as well, meaning that cooling is applied on the hot side.

For the heat rate source component, the heat rate is fully applied on port p2.

For the thermoelectric module component, the heat rates $P_{1,src}$ and $P_{2,src}$ can be expressed in different ways. See [Theory for the Thermoelectric Module Component](#) for more details.

Theory for the Conductive Thermal Resistor Component

This section presents the underlying theory of the [Events Timeline](#) 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

$$P = -\frac{\Delta T}{R} \quad (4-90)$$

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

The expression of the thermal resistance used in [Equation 4-90](#) 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 4-90](#) 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

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

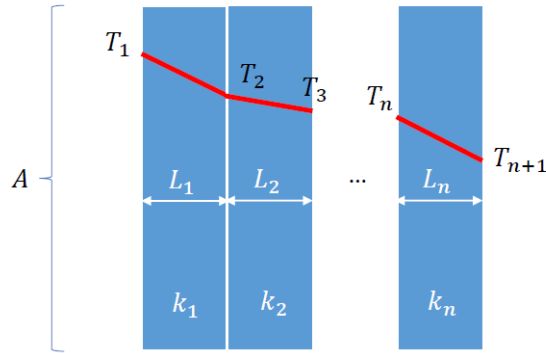
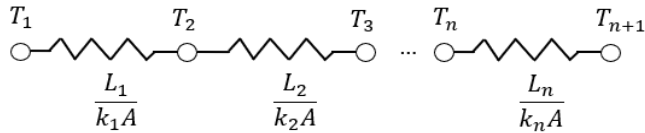


Figure 4-7: Steady conduction through a composite wall made of n layers.

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

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

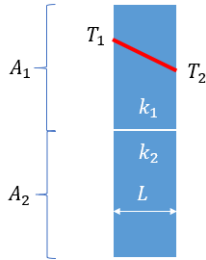
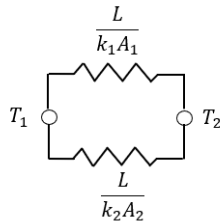


Figure 4-8: Steady conduction through a composite wall made of n slabs.

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

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:

$$R_i = \frac{1 - \epsilon_i}{A_i \epsilon_i}$$

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:

$$R_{12} = \frac{1 - \epsilon_1}{A_1 \epsilon_1} + \frac{1}{A_1 F_{12}} + \frac{1 - \epsilon_2}{A_2 \epsilon_2}$$

And the net radiation exchange is expressed as:

$$P_{12} = \frac{\sigma(T_1^4 - T_2^4)}{R_{12}}$$

For specific configurations, the surface areas can be evaluated from the geometric dimensions and the view factors are equal to 1. Therefore the expression above simplifies (see [Ref. 21](#) for details).

Concentric Spheres

The surface areas A_1 and A_2 satisfy

$$\frac{A_1}{A_2} = \left(\frac{r_1}{r_2}\right)^2$$

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 satisfy

$$A_1 = A_2$$

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 satisfy

$$\frac{A_1}{A_2} = \frac{r_1}{r_2}$$

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

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: m³), ρ the density (SI unit: kg/m³), C_p the heat capacity at constant pressure (SI unit: J/(kg·K)), and m the mass (SI unit: kg).

LUMPED THERMAL CAPACITANCE MODEL

Under certain conditions, the temperature T of a solid body subjected to convective heat transfer on its boundaries may be considered as homogeneous, depending only on time, using the thermal capacitance:

$$\frac{T - T_\infty}{T_0 - T_\infty} = e^{-\frac{hA}{C}t}$$

where:

- T_0 is the initial body temperature
- T_∞ is the surrounding fluid temperature
- h is the convective heat transfer coefficient (SI unit: W/(m²·K))
- A is the surface of convective heat transfer (SI unit: m²)

This approximation, referred as the lumped thermal capacitance model (see [Ref. 41](#)), holds when h , C , and A are constant and the gradients of temperature within the body are expected to be smaller than the gradients of temperature between the body and the surrounding. It happens for example when the thermal contact between the solid and the fluid is poor, or when the solid is a good thermal conductor.

This is characterized by the Biot number Bi (dimensionless):

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

where L is a relevant length scale of the body and k is its thermal conductivity.

The lumped thermal capacitance model is usually assumed to be valid when $Bi < 0.1$.

Regarding the network representation of thermal systems, the Biot number may be used to determine how many nodes should be included, assuming a 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.

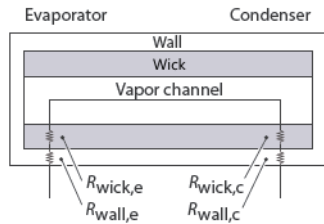


Figure 4-9: Heat pipe components.

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:

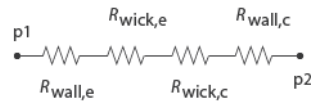


Figure 4-10: Thermal network representation of a heat pipe.

See [Ref. 18](#) for details about the network representation of heat pipes and the expressions of effective thermal conductivity of the wick used to express its thermal resistance.

Theory for the Thermoelectric Module Component

Thermoelectric modules are devices which use thermoelectricity (Peltier effect) to transfer heat from a source to a sink.

DEVICE DESCRIPTION

They are composed of thermocouples, each consisting of p-type and n-type semiconductors, connected electrically in series and thermally in parallel and sandwiched between two high thermally conductive but low electrically conductive ceramic plates, as described on [Figure 4-11](#) below.

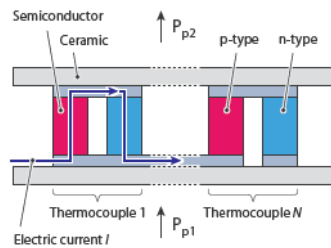


Figure 4-11: Thermoelectric module schematic description.

See [Theory for the Thermoelectric Effect Interface](#) for details about the Peltier effect.

HEAT RATE

In heating and cooling operating modes of the thermoelectric module (that is, no heat generation), the heat rates at the cold and hot junctions of each thermocouple are due to Peltier effect, Joule heating, thermal conduction, and heat storage:

- Hot junction:

$$P_{\text{hot}} = SIT_{\text{hot}} + \frac{R_e I^2}{2} - \frac{\Delta T}{R} - C_{\text{hot}} \frac{\partial T_{\text{hot}}}{\partial t} \quad (4-91)$$

- Cold junction:

$$P_{\text{cold}} = -SIT_{\text{cold}} + \frac{R_e I^2}{2} + \frac{\Delta T}{R} - C_{\text{cold}} \frac{\partial T_{\text{cold}}}{\partial t} \quad (4-92)$$

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 electric 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 4-91](#) and [Equation 4-92](#) the assumption of a symmetric distribution of Joule heating at the hot and cold junctions has been made.

THERMOCOUPLE ELECTRIC AND THERMAL RESISTANCES

As the p-type and n-type semiconductors are connected electrically in series, the electric 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 electric resistances of the p-type and n-type semiconductors.

And as they are connected thermally in parallel, the thermal resistance R of each thermocouple is expressed as

$$\frac{1}{R} = \frac{1}{R_p} + \frac{1}{R_n}$$

where R_p and R_n (SI unit: K/W) are the thermal resistances of the p-type and n-type semiconductors.

THERMOCOUPLE SEEBECK COEFFICIENT

Finally, the Seebeck coefficient S of each thermocouple is expressed as

$$S = S_p - S_n$$

TOTAL HEAT RATE

The total heat rate through the module is obtained by summing the heat rates in each thermocouples.

NETWORK REPRESENTATION

The following network representation corresponds to [Equation 4-91](#) and [Equation 4-92](#):

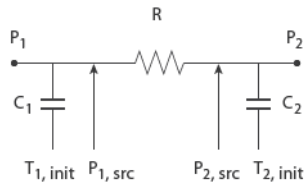


Figure 4-12: Thermal network representation of a thermoelectric module.

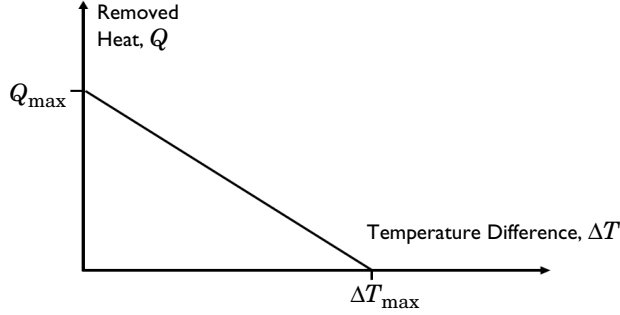
See [Ref. 18](#) and [Ref. 19](#) for details about the network representation of thermoelectric modules.

PERFORMANCE GRAPHS FOR THERMOELECTRIC COOLERS

It is usual to characterize thermoelectric coolers through graphs showing the heat removed from the cold side of the module, Q , in function of the difference of temperature ΔT between the hot and cold sides, the input intensity I , and the ambient temperature.

Linearized Model at Given Intensity and Ambient Temperature

This model establishes a linear relation between the removed heat and the temperature difference, as shown in 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 Cell Periodicity

This section presents the underlying theory of the [Cell Periodicity](#) feature, and the resulting heat transfer equations that hold.

The heterogeneous properties of a material at the microscopic scale are often impractical for direct application at the macroscopic scale. In such cases, it becomes necessary to employ a homogeneous material model that incorporates appropriately equivalent properties.

The [Cell Periodicity](#) feature facilitates the evaluation of such average properties for heat transfer. It is based on the idea of a repeating unit cell (RUC) or a representative volume element (RVE). The cell is a microscopic domain that is representative for the material on a macroscopic length scale.

The terms RVE (Representative Volume Element) and RUC (Repeating Unit Cell) are frequently used interchangeably in the academic literature. Generally, an RVE describes a subvolume that characterizes heterogeneous materials with a statistically homogeneous microstructure. Conversely, the RUC is employed to represent subvolumes characterized by periodic patterns.

When dealing with materials characterized by a random distribution of particles, fibers, or pores, the appropriate subvolume to consider would be an RVE. To be numerically efficient, this RVE should be as small as possible, but sufficiently large to provide a representative sample for the determination of average material properties at a macroscopic scale.

If the material or lattice exhibits true periodicity, the suitable choice for representing a material subvolume would be an RUC, which repeats itself to uniformly fill the space with a repetitive pattern.

The distinction between RVEs and RUCs necessitates the application of different sets of boundary conditions. The **Cell Periodicity** node offers two distinct sets of boundary conditions, namely **Uniform temperature gradient** and **Periodic temperature**, designed to suit these subvolumes respectively.

HOMOGENIZATION METHOD

The homogenization method introduces two scales: a macroscale and a microscale. The macroscale usually refers to the homogenized continuous medium, and the microscale to the heterogeneous cell. The macroscopic conductive heat flux $\mathbf{q}_{\text{avg}}$ and

the macroscopic temperature gradient ∇T_{avg} are derived by averaging the conductive heat flux and temperature gradient in the cell:

$$\mathbf{q}_{\text{avg}} = \frac{1}{V} \int_V \mathbf{q} dV \text{ and } \nabla T_{\text{avg}} = \frac{1}{V} \int_V \nabla T dV \quad (4-93)$$

where V is the volume of the cell. The macroscopic thermal conductivity tensor $\bar{\mathbf{k}}$ of the homogenized continuum is then defined by

$$\mathbf{q}_{\text{avg}} = -\bar{\mathbf{k}} \nabla T_{\text{avg}} \quad (4-94)$$

UNIFORM TEMPERATURE GRADIENT BOUNDARY CONDITION

Let's consider the temperature field on the outer boundaries of a RVE written as:

$$T = \nabla T_0 \cdot \mathbf{X}$$

where ∇T_0 is a uniform temperature gradient, and $\mathbf{X}$ is the position. By applying the divergence theorem, it can be demonstrated that:

$$\nabla T_{\text{avg}} = \nabla T_0$$

The volume-averaged temperature gradient in the RVE is then equal to the uniform temperature gradient applied on its outer boundaries, irrespective of the constitution of the material subvolume.

PERIODIC TEMPERATURE BOUNDARY CONDITIONS

For a periodic structure that consists of an array of RUC, the temperature field over cell outer boundaries is written as:

$$T = \nabla T_0 \cdot \mathbf{X} + T^*$$

where ∇T_0 is a uniform average temperature gradient, $\mathbf{X}$ is the position, and T^* is a function that is periodic from one cell to another. Given that the array of cells forms a continuous structure, it is essential to ensure continuity of temperature across the boundaries between these cells. The boundaries in a **Boundary Pair** subnode must therefore always appear in parallel pairs. One of them is labeled as *source* and the other as *destination*. The temperature on a pair of parallel (and opposite) boundaries can be written as:

$$T_{\text{dst}} = \nabla T_0 \cdot \mathbf{X}_{\text{dst}} + T^* \text{ and } T_{\text{src}} = \nabla T_0 \cdot \mathbf{X}_{\text{src}} + T^*$$

where $\mathbf{X}_{\text{dst}}$ and T_{dst} are the position and temperature on the destination side, and $\mathbf{X}_{\text{src}}$ and T_{src} are the position and temperature on the source side.

Hence, the temperature boundary condition is established by:

$$T_{\text{dst}} - T_{\text{src}} = \nabla T_0 \cdot (\mathbf{X}_{\text{dst}} - \mathbf{X}_{\text{src}})$$

As for the uniform temperature gradient boundary condition, applying the divergence theorem leads to:

$$\nabla T_{\text{avg}} = \nabla T_0$$

The volume-averaged temperature gradient in the RUC is also equal to the uniform temperature gradient applied on its outer boundaries, irrespective of the constitution of the material subvolume. In that case, it is required to constrain the temperature at one point of the RUC to ensure uniqueness of the solution.

PRESCRIBED AVERAGE TEMPERATURE GRADIENT

In order to determine the homogeneous thermal conductivity tensor of a heterogeneous material, the material is subjected on its outer boundaries to a non-zero uniform temperature gradient ∇T_0 in each direction, one-by-one, while keeping the other components equal to zero. With both uniform temperature gradient and periodic temperature boundary conditions, the global average temperature gradient ∇T_{avg} is equal to this uniform temperature gradient ∇T_0 and is then a user input. The average conductive heat flux $\mathbf{q}_{\text{avg}}$ is computed from [Equation 4-93](#), and the thermal conductivity tensor $\bar{k}$ is obtained from [Equation 4-94](#).

Theory for Thermal Connection

This section presents the underlying theory of the [Thermal Connection](#) and [Thermal Connection, Interface-Interface](#) features, and the resulting heat transfer equations that hold.

In this section:

- [Conductive Thermal Resistor](#)
- [Thermal Capacitor](#)

Conductive Thermal Resistor

This section presents the underlying theory of the **Conductive thermal resistor** option.

THERMAL RESISTANCE ANALOGY FOR STEADY CONDUCTION

By analogy with Ohm's law for electric current, the heat flux q (SI unit: W/m^2) in a material due to the temperature difference ΔT (SI unit: K) is

$$-\mathbf{n} \cdot \mathbf{q} = \frac{\Delta T}{A_1 R} \text{ on the selection of connector 1} \quad (4-95)$$

$$-\mathbf{n} \cdot \mathbf{q} = -\frac{\Delta T}{A_2 R} \text{ on the selection of connector 2} \quad (4-96)$$

where R (SI unit: K/W) is the thermal resistance, A_1 (SI unit: m^2) is the area of the selection of connector 1 and A_2 (SI unit: m^2) is the area of the selection of connector 2.

The expression of the thermal resistance used in [Equation 4-95](#) and [Equation 4-96](#) depends on the geometric configuration.

Plane Shell

When considering steady conduction through a plane shell of surface area A (SI unit: m^2), thickness L (SI unit: m), and constant thermal conductivity k (SI unit: $\text{W}/(\text{m} \cdot \text{K})$), 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 4-90 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):

$$-\mathbf{n} \cdot \mathbf{q} = \frac{\Delta T_{\text{overall}}}{A_1 R_{\text{tot}}} \text{ on the selection of connector 1}$$

$$-\mathbf{n} \cdot \mathbf{q} = -\frac{\Delta T_{\text{overall}}}{A_2 R_{\text{tot}}} \text{ on the selection of connector 2}$$

where R_{tot} (SI unit: K/W) is the total thermal resistance, A_1 (SI unit: m²) is the area of the selection of connector 1 and A_2 (SI unit: m²) is the area of the selection of connector 2.

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

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

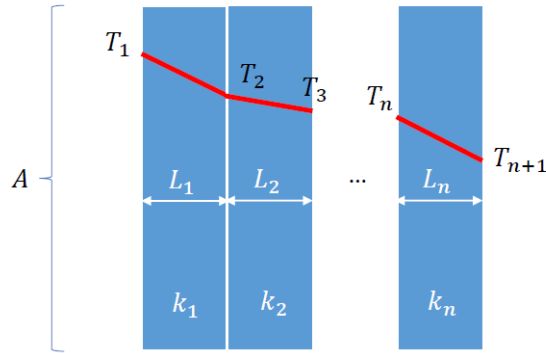
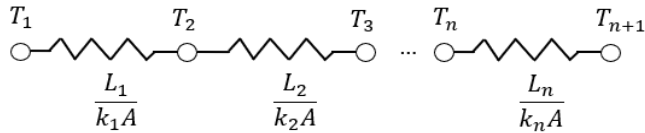


Figure 4-13: Steady conduction through a composite wall made of n layers.

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

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

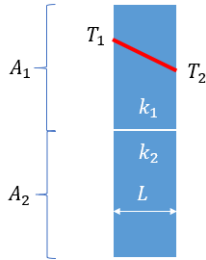
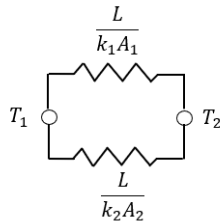


Figure 4-14: Steady conduction through a composite wall made of n slabs.

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.

Thermal Capacitor

This section presents the underlying theory of the **Thermal capacitor** option.

THERMAL CAPACITANCE

The heat flux q (SI unit: W/m^2) in a material due to the temperature difference ΔT (SI unit: K) is:

$$-\mathbf{n} \cdot \mathbf{q} = \frac{C}{A_1} \frac{d(\Delta T)}{dt} \text{ on the selection of connector 1} \quad (4-97)$$

$$-\mathbf{n} \cdot \mathbf{q} = -\frac{C}{A_2} \frac{d(\Delta T)}{dt} \text{ on the selection of connector 2} \quad (4-98)$$

where C (SI unit: J/K) is the thermal capacitance. The thermal capacitance 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 pressure (SI unit: J/(kg·K)), and m the mass (SI unit: kg).

LUMPED THERMAL CAPACITANCE MODEL

Under certain conditions, the temperature T of a solid body subjected to convective heat transfer on its boundaries may be considered as homogeneous, depending only on time, using the thermal capacitance:

$$\frac{T - T_\infty}{T_0 - T_\infty} = e^{-\frac{hA}{C}t}$$

where:

- T_0 is the initial body temperature
- T_∞ is the surrounding fluid temperature
- h is the convective heat transfer coefficient (SI unit: $W/(m^2 \cdot K)$)
- A is the surface of convective heat transfer (SI unit: m^2)

This approximation, referred as the lumped thermal capacitance model (see [Ref. 41](#)), holds when h , C , and A are constant and the gradients of temperature within the body are expected to be smaller than the gradients of temperature between the body and the surrounding. It happens for example when the thermal contact between the solid and the fluid is poor, or when the solid is a good thermal conductor.

This is characterized by the Biot number Bi (dimensionless):

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

where L is a relevant length scale of the body, and k is its thermal conductivity.

The lumped thermal capacitance model is usually assumed to be valid when $\text{Bi} < 0.1$.

Regarding the network representation of thermal systems, the Biot number may be used to determine how many nodes should be included, assuming a homogeneous temperature distribution in the corresponding domains.

Theory for Surface-to-Surface Radiation

In addition to conduction and convection, the third mechanism for heat transfer is radiation. Consider an environment with fully transparent or fully opaque objects. Thermal radiation denotes the stream of electromagnetic waves emitted from a body at a certain temperature.

The Surface-to-Surface Radiation Interface theory is described in this section:

- Deriving the Radiative Heat Flux for Opaque Surfaces
- Deriving the Radiative Heat Flux for Semitransparent Surfaces
- Wavelength Dependence of Surface Properties
- Directional Dependence of Surface Properties
- The Radiosity Method for Diffuse-Gray Surfaces
- The Radiosity Method for Diffuse-Spectral Surfaces
- View Factor Evaluation

Deriving the Radiative Heat Flux for Opaque Surfaces

In Figure 4-15, 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.

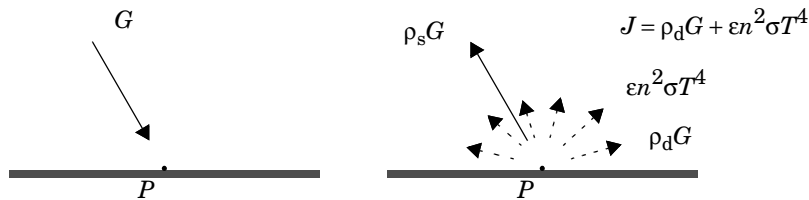


Figure 4-15: Incoming irradiation (left), outgoing radiosity (right).

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:

$$J = \rho_d G + \varepsilon e_b(T) \quad (4-99)$$

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

$$q = (1 - \rho_s)G - J \quad (4-100)$$

Using Equation 4-99 and Equation 4-100, J can be eliminated and a general expression is obtained for the net inward heat flux into the opaque body based on G and T .

$$q = (1 - (\rho_d + \rho_s))G - \varepsilon e_b(T) \quad (4-101)$$

Most opaque bodies also behave as ideal gray bodies, meaning that the absorptivity and emissivity are equal, and the reflectivity $\rho_d + \rho_s$ is therefore obtained from the following relation:

$$\alpha = \varepsilon = 1 - (\rho_d + \rho_s) \quad (4-102)$$

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

$$q = \varepsilon(G - e_b(T)) \quad (4-103)$$

This is the expression used for the radiative boundary condition.

Deriving the Radiative Heat Flux for Semitransparent Surfaces

In Figure 4-15, 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.

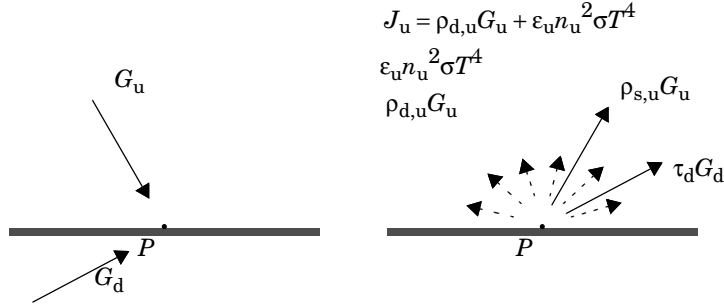


Figure 4-16: Upside and downside incoming irradiation (left), upside outgoing radiosity (right). The downside outgoing radiosity is defined in a similar way.

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 and emitted radiation:

$$J_u = \rho_{d,u} G_u + \epsilon_u e_{b,u}(T_u) \quad (4-104)$$

$$J_d = \rho_{d,d} G_d + \epsilon_d e_{b,d}(T_d) \quad (4-105)$$

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 outgoing radiation (radiosity, specular reflected and transmitted radiations):

$$q_u = (1 - \rho_{s,u} - \tau_u) G_u - J_u \quad (4-106)$$

$$q_d = (1 - \rho_{s,d} - \tau_d) G_d - J_d \quad (4-107)$$

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:

$$\epsilon_u + \rho_{d,u} = 1 - \rho_{s,u} - \tau_u \quad (4-108)$$

$$\epsilon_d + \rho_{d,d} = 1 - \rho_{s,d} - \tau_d \quad (4-109)$$

Using Equation 4-104 to Equation 4-109, 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 :

$$q_u = \varepsilon_u(G_u - e_{b,u}(T_u)) \quad (4-110)$$

$$q_d = \varepsilon_d(G_d - e_{b,d}(T_d)) \quad (4-111)$$

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

$$q = \varepsilon_u(G_u - e_{b,u}(T_u)) + \varepsilon_d(G_d - e_{b,d}(T_d)) \quad (4-112)$$

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 Properties

The surface properties for radiation, the emissivity, specular reflectivity and specular transmissivity can be dependent on the incident angle, the surface temperature, or the radiation wavelength. The emissivity, specular reflectivity and specular transmissivity are defined in Ref. 20.

The Surface-to-Surface Radiation interface in the Heat Transfer module implements the radiosity method that enables arbitrary temperature and wavelength dependent surface properties. The dependence of the surface properties on the incident angle can be considered by using the ray shooting method ([Directional Dependence of Surface Properties](#)).

PLANCK SPECTRAL DISTRIBUTION

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

The blackbody hemispherical emissive power (SI unit: $\text{W}/(\text{m}^2 \cdot \text{sr})$), is denoted $e_{b,\lambda}(\lambda, T)$, and defined as (1-37 in [Ref. 20](#)):

$$e_{b,\lambda}(\lambda, T) = \frac{2\pi n^2 C_1}{\lambda^5 \left(e^{\frac{C_2}{\lambda T}} - 1 \right)} \quad (4-113)$$

where:

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

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

- h is the Planck constant (SI unit: $\text{J} \cdot \text{s}$)
- 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 4-17 and Figure 4-18 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).

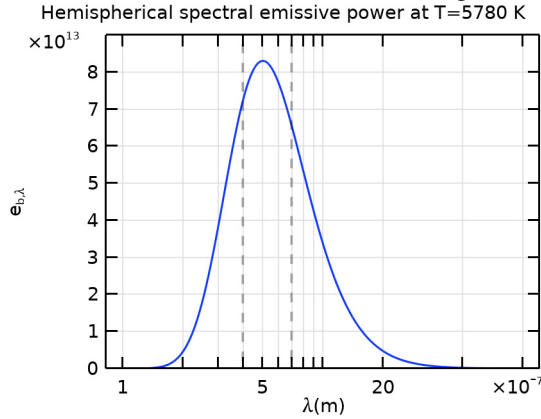


Figure 4-17: Planck distribution of a blackbody at 5780 K.

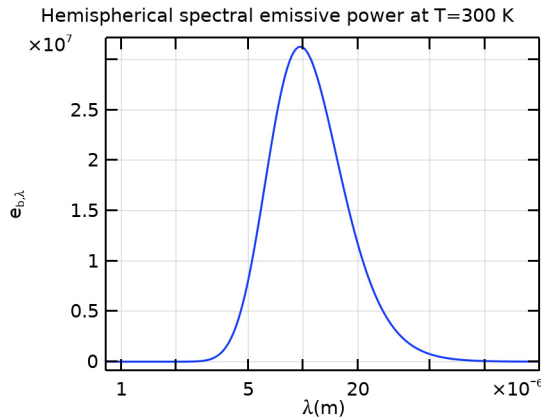


Figure 4-18: Planck distribution of a blackbody at 300 K.

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

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

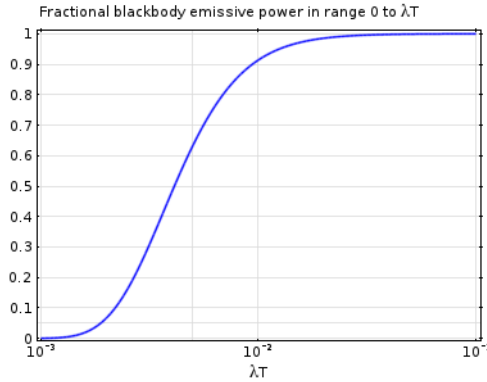


The function $e_b(T)$ is available as a predefined function via `rad.feb(T)` in the Surface-to-Surface Radiation interface.

Notice that

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

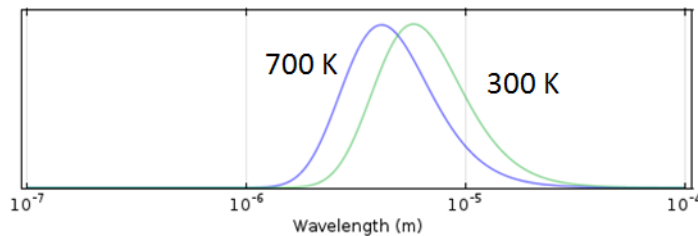
The figure below shows the value of $F_{0 \rightarrow \lambda T}$ for different values of λT .



GRAY SURFACES

Gray surfaces correspond to the hypothesis that surface properties are independent of the radiation wavelength. This corresponds to Constant wavelength dependence in the Surface-to-Surface Radiation interface.

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



This setting is rarely applicable if there is solar radiation.

SOLAR AND AMBIENT SPECTRAL BANDS

When solar radiation is part of the model, it is possible to enhance a diffuse-gray surface model by considering two spectral bands: one for short wavelengths and one for large wavelengths. This corresponds to Solar and Ambient wavelength dependence in the Surface-to-Surface Radiation interface.

It is interesting to notice that about 97% of the radiated power from a blackbody at 5800 K is at wavelengths of $2.5 \mu\text{m}$ 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 4-19](#)).

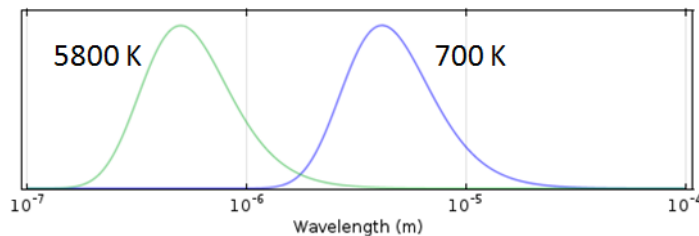


Figure 4-19: Normalized Planck distribution of blackbodies at 700 K and 5800 K.

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.

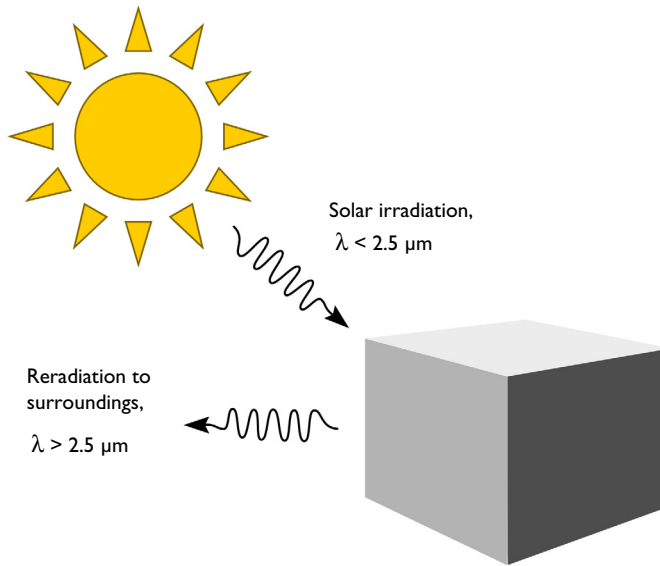


Figure 4-20: Absorption of solar radiation and emission to the surroundings.

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.

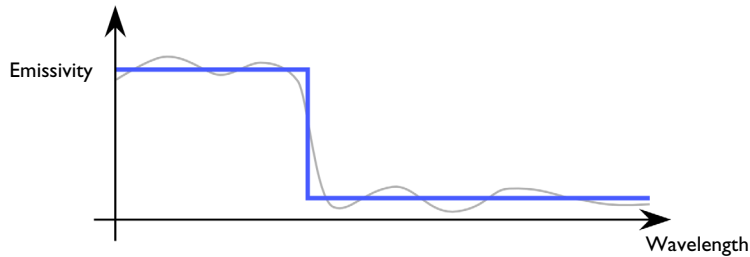


Figure 4-21: Solar and ambient spectral band approximation of the surface emissivity by a constant per band emissivity.

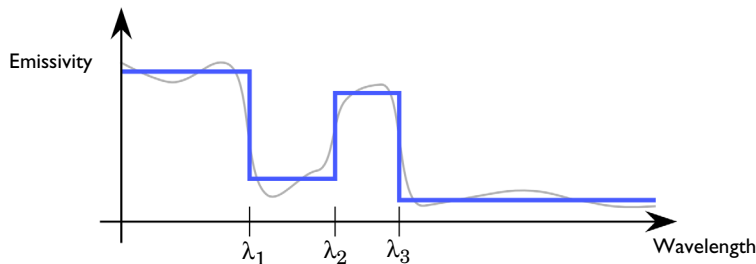


Continuous and piecewise constant predefined functions are available, as well as a predefined plot, in the **Surface-to-Surface Radiation** interface to display the wavelength dependency of material properties in the manner of the figure above. See [Predefined Functions for the Surface-to-Surface Radiation Interface](#) for details.

GENERAL SPECTRAL SURFACES

Spectral surfaces correspond to the hypothesis that surface properties are wavelength dependent. This corresponds to Multiple spectral bands wavelength dependence in the Surface-to-Surface Radiation interface.

The Heat Transfer Module supports constant surface properties per spectral bands and to adjust spectral intervals endpoints.



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

Directional Dependence of Surface Properties

The Surface-to-Surface Radiation interface in the Heat Transfer module implements the ray shooting method, that enables to set the dependence of the emissivity, of the specular reflectivity and of the specular transmissivity on the incident angle.

The polar and azimuthal angles, $\theta \in [0, \pi/2]$ and $\varphi \in [-\pi, \pi]$, are defined below:

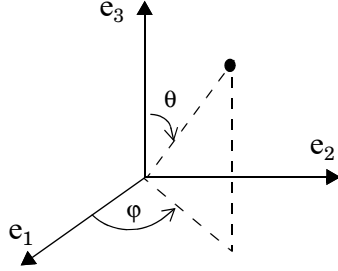


Figure 4-22: Definition of polar and azimuthal angles.

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

- Emissivity

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

- Transmissivity

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

- Specular reflectivity

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

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 4-100, this source account for the difference between incident radiation, or *irradiation*, G , and radiation leaving the surface, or *radiosity*, J :

$$q = (1 - \rho_s)G - J$$

The radiosity, J , is given in Equation 4-99. 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:

$$G = G_m + G_{\text{ext}} + G_{\text{amb}} \quad (4-114)$$

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:

$$J = \rho_{\text{d}}(G_{\text{m}}(J) + G_{\text{ext}} + G_{\text{amb}}) + \epsilon e_{\text{b}}(T) \quad (4-115)$$

- [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. 21](#)). It resembles a mirror that absorbs all irradiation and then radiates it back in all directions.
- [Prescribed Radiosity \(Surface-to-Surface Radiation Interface\)](#) makes it possible to specify *graybody radiation*. The radiosity expression is then $\epsilon e_{\text{b}}(T)$. 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 4-115](#).

- [Specular Mirror \(Surface-to-Surface Radiation Interface\)](#) is a variant of the Opaque Surface radiation type with $\epsilon = \rho_{\text{d}} = 0$. Following the conservation equation, the specular surface reflectivity is then defined as $\rho_{\text{s}} = 1$. Zero values of emissivity and diffuse reflectivity result in identically zero radiosity.

- **Semitransparent Surface (Surface-to-Surface Radiation Interface)** is available when the surface-to-surface radiation method is Ray shooting. It accounts for reflection and transmission; the conservation equation reads

$$\varepsilon + \rho_d + \rho_s + \tau = 1,$$

and the radiosities read

$$J_u = (\rho_{d,u}(G_{m,u}(J_u) + G_{ext,u} + G_{amb,u}) + \varepsilon_u e_b(T)) + \tau_d(G_{m,d}(J_d) + G_{ext,d} + G_{amb,d}) \quad (4-116)$$

$$J_d = (\rho_{d,d}(G_{m,d}(J_d) + G_{ext,d} + G_{amb,d}) + \varepsilon_d e_b(T)) + \tau_d(G_{m,u}(J_u) + G_{ext,u} + G_{amb,u}) \quad (4-117)$$

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 4-113](#).

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:

$$J_i = \rho_{d, i}(G_{m, i}(J_i) + G_{\text{ext}, i} + G_{\text{amb}, i}) + \varepsilon_i e_b(T) \quad (4-118)$$

- **Diffuse Mirror (Surface-to-Surface Radiation Interface)** is a variant of the Diffuse Surface radiation type with $\varepsilon_i = 0$. Reradiation surfaces are common as an approximation of a surface that is well insulated on one side and for which convection effects can be neglected on the opposite (radiating) side (see [Ref. 21](#)). It resembles a mirror that absorbs all irradiation and then radiates it back in all directions.
- **Prescribed Radiosity (Surface-to-Surface Radiation Interface)** makes it possible to specify the surface radiation for each spectral band. Using the graybody radiation definition, the radiosity is then $F_{\lambda_1 T \rightarrow \lambda_2 T} e_b(T)$. 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 4-118](#).

- **Specular Mirror (Surface-to-Surface Radiation Interface)** is a variant of the Opaque Surface radiation type with $\varepsilon_i = \rho_{d, i} = 0$. Following the conservation equation, the specular surface reflectivity is then defined as $\rho_{s, i} = 1$. Zero values of emissivity and diffuse reflectivity result in identically zero radiosity.
- **Semitransparent Surface (Surface-to-Surface Radiation Interface)** is available with Ray shooting as Surface-to-surface radiation method. It accounts for reflection and transmission; the conservation equation reads

$$\varepsilon_i + \rho_{d,i} + \rho_{s,i} + \tau_i = 1$$

and the radiosities read

$$J_{u,i} = \rho_{d,u,i}(G_{mu,i}(J_{u,i}) + G_{ext,u,i} + G_{amb,u,i}) + \varepsilon_{u,i}e_b(T) + \tau_{d,i}(G_{md,i}(J_{d,i}) + G_{ext,d,i} + G_{amb,d,i}) \quad (4-119)$$

$$J_{d,i} = \rho_{d,u,i}(G_{md,i}(J_{d,i}) + G_{ext,d,i} + G_{amb,d,i}) + \varepsilon_{d,i}e_b(T) + \tau_{u,i}(G_{mu,i}(J_{u,i}) + G_{ext,u,i} + G_{amb,u,i}) \quad (4-120)$$

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 4-115 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 4-23. 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} .

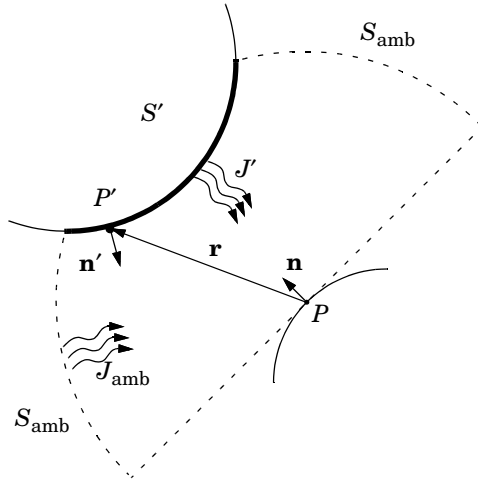


Figure 4-23: Example geometry for surface-to-surface radiation.

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 4-114 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

$$G_m = \int_{S_{\perp}'} \frac{(-\mathbf{n}' \cdot \mathbf{r}_{\perp})(\mathbf{n} \cdot \mathbf{r}_{\perp})}{2|\mathbf{r}_{\perp}|^3} J' ds \quad (4-121)$$

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

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 4-121](#). For cases with specularly reflective surfaces, specular view factors depends also on specular reflectivities of surrounding surfaces, see [Ref. 23](#).

A separate evaluation is performed for each unique point where G_m or F_{amb} is requested, typically for each quadrature point during solution. Differential view factors are normally computed only once, the first time they are needed, and then stored in memory until next time the model definition or the mesh is changed.

The Heat Transfer Module supports the following surface-to-surface radiation methods, which are selected in the Radiation Settings section in a Heat Transfer interface:

- Hemicube
- Direct area integration
- Ray shooting



View factors are always calculated directly from the mesh, which is a polygonal representation of the geometry. To improve the accuracy of the radiative heat transfer simulation, the mesh must be refined rather than raising the element order.

VIEW FACTOR FOR EXTERNAL RADIATION SOURCES

In 3D, the view factor for a point at finite distance is given by

$$\frac{\cos \theta}{4\pi r^2}$$

where θ is the angle between the normal to the irradiated surface and the direction of the source, and r is the distance from the source. For a source at infinity, the view factor is given by $\cos \theta$.

In 2D the view factor for a point at finite distance is given by

$$\frac{\cos \theta}{2\pi r}$$

and the view factor for a source at infinity is $\cos \theta$.

SOLAR POSITION

The Sun is the most common example of an external radiation source. The position of the Sun is necessary to determine the direction of the corresponding external radiation source. The direction of sunlight (zenith angle and the solar elevation) is automatically computed from the latitude, longitude, time zone, date, and time using similar a method as described in [Ref. 20](#). The estimated solar position is accurate for a date between year 2000 and 2199, due to an approximation used in the Julian Day calendar calculation.

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

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

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, the ray shooting 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.

FLUENCE RATE

The fluence rate is evaluated at a point P , as in [Figure 4-23](#), using the following surface integral:

$$E_0 = \int_{S'} \frac{J'}{\pi |\mathbf{r}|^2} ds \quad (4-122)$$

where S' corresponds to all the surfaces which can be seen from point P . J' is the radiosity emitted by the surface S' in P direction. $\mathbf{r}$ is the vector which points from P to P' located on S' .

For the 2D case, the resulting equations for the fluence rate simplifies to:

$$E_0 = \int_{S_{\perp}'} \frac{J'}{2|\mathbf{r}_{\perp}|} ds \quad (4-123)$$

where the integral over $S_{\perp}'$ denotes the line integral along the boundaries of the 2D geometry.

Theory for Orbital Thermal Loads

The [Orbital Thermal Loads Interface](#) theory is described in this section:

- [Coordinate Systems](#)
- [Defining the Position of the Spacecraft Over Time](#)
- [Orienting the Spacecraft](#)
- [Modeling the Planet](#)
- [Modeling of Fast Tumbling](#)

Coordinate Systems

One of the first requirement to describe a spacecraft in orbit is to define the different coordinate systems to work with:

- The equatorial coordinate system (ECS) has its origin at the center of the planet and its first axis pointing toward the Vernal point on the celestial sphere. The third axis is parallel to the rotation of the planet, pointing toward North. The second axis completes a right-handed coordinate system.
- The planetary coordinate system (PCS) is a spherical coordinate system, in terms of latitude and longitude with the same center as the ECS. It is assumed that planets are perfect spheres. The cartesian PCS can also be used, with the first axis pointing to longitude 0, latitude 0, the second axis pointing to longitude +90, latitude 0, and the third axis to the North pole (parallel to the third axis of the ECS). It is rotating in time about the third axis related to the ECS.
- The spacecraft coordinate system (SCS) is a cartesian coordinate system with its origin at the location of the spacecraft in orbit. The primary and secondary axis are defined by the user in the [Spacecraft Orientation](#) feature. The secondary axis is defined as the projection of the user-specified direction onto the plane perpendicular to the primary axis. The third axis completes the right-handed coordinate system. The selection of these axis can change during orbit.
- The geometry coordinate system is the material frame, where the model exists. All vectors describing the thermal loads are to be transformed into this system.
- The orbital plane coordinate system is parallel to the orbital plane. The first axis points from the planet center to the periapsis and the spacecraft rotates about the third axis, which is normal to the plane of orbit. The elliptical orbit is computed in

this plane, and then rotated into the ECS. The orbital plane coordinate system is not defined with **User defined** selected as **Orbit type** in the **Orbital Parameters** feature.

Defining the Position of the Spacecraft Over Time

With any other option than **User defined** selected as **Orbit type** in the **Orbital Parameters**, the orbit is computed using the six Keplerian elements as in [Figure 4-24](#): the eccentricity (e), argument of periapsis (ω), semi-major axis (a), inclination, (i) and the longitude of the ascending node, (Ω). The beginning of the simulation can start at any point on this ellipse, as defined by the true anomaly at the simulation start time (v_0).

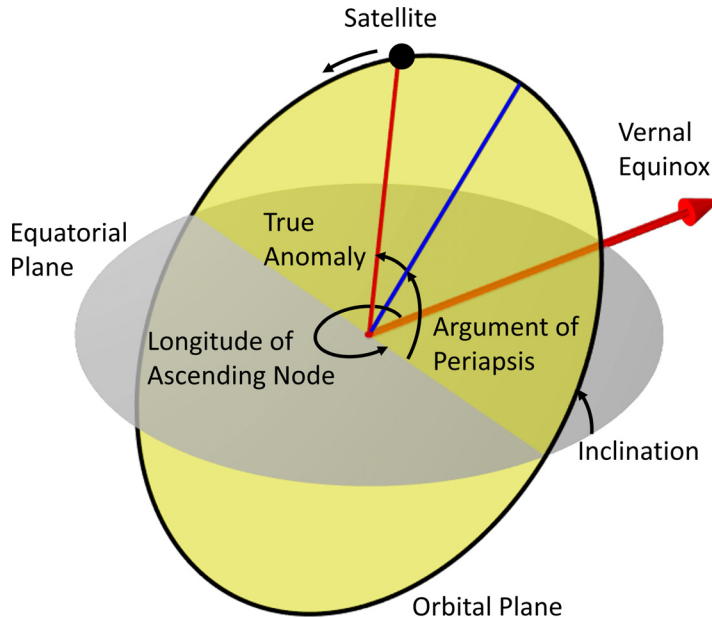


Figure 4-24: Definition of Keplerian elements.

It is also necessary to orient the planet at the initial time of the simulation, for the cases where the albedo or the planet radiative flux are a function of longitude and latitude, and for planet-position based orbital maneuvers. See **Planet longitude at start time** in the [Planet Properties](#) feature for details about the available options.

Now that the orbit is completely defined, and the position of the planet is defined, we move on to computing the spacecraft position over time.

In the orbital plane coordinate system, the coordinates of the spacecraft over time describe an ellipse. We use the solutions to Kepler's laws to solve for the eccentric anomaly, E , and true anomaly, v , by starting with the mean anomaly:

$$M = 2\pi \frac{(t + t_0)}{T}$$

where the orbital period is $T = 2\pi\sqrt{a^3/(GM_{\text{planet}})}$ and t_0 is the time that it takes for the spacecraft to get from the perigee to v_0 . This time is computed by integrating the swept area of the ellipse in cylindrical coordinates, centered at the planet:

$$t_0 = \frac{T}{A} \int_0^{v_0} \int_0^{r(\theta)} r(dr) d\theta$$

where A is the total area of the orbital ellipse and the ellipse radius as a function of the true anomaly is:

$$r(v) = \frac{a(1-e^2)}{1+e\cos v}$$

Now that we can compute the mean anomaly over time, we use a series expansion to get the eccentric anomaly:

$$E = M + 2 \sum_{k=1}^{\infty} \frac{1}{k} J_k(ke) \sin(kM)$$

and the true anomaly:

$$v = M + 2 \sum_{k=1}^{\infty} \frac{1}{k} \left(J_k(ke) + \sum_{p=1}^{\infty} \beta^p (J_{k-p}(ke) + J_{k+p}(ke)) \right) \sin(kM)$$

where $\beta = (1 - \sqrt{1-e^2})/e$ and J_n are Bessel functions. In practice, only a finite number of terms, depending upon the eccentricity, in this infinite series need to be considered.

The number of terms for the series is

$$N_{\text{terms}} = \begin{cases} 0 & \text{for } e = 0 \\ [4 + 20e] & \text{for } 0 < e \leq 0.5 \\ [4 + 20e + 20e - 0.5^2] & \text{for } e > 0.5 \end{cases}$$

where $[x]$ is the function that rounds to closest integer or to closest number.

Once the true anomaly over time is known, we know the position, $\mathbf{X}_{ECS}$, in the ECS from the longitude of ascending node, the inclination, and the argument of periapsis:

$$\mathbf{X}_{ECS} = \begin{bmatrix} x_{ECS} \\ y_{ECS} \\ z_{ECS} \end{bmatrix} = T_{ECS} \begin{bmatrix} r(v) \cos v \\ r(v) \sin v \\ 0 \end{bmatrix}$$

where T_{ECS} is the transformation matrix from the orbital plane coordinate system to ECS:

$$T_{ECS} = \begin{bmatrix} \cos \Omega \cos \omega - \cos i \sin \Omega \sin \omega & -\cos \Omega \sin \omega - \cos i \cos \Omega \sin \Omega & \sin \Omega \sin i \\ \cos \omega \sin \Omega + \cos \Omega \cos i \sin \omega & \cos \Omega \cos i \cos \omega - \sin \Omega \sin \omega & -\cos \Omega \sin i \\ \sin i \sin \omega & \sin i \cos \omega & \cos i \end{bmatrix}$$

To compute the times when the spacecraft goes into and out of eclipse, we introduce the concept of the solar normal plane (SNP) that is perpendicular to the Sun vector, $\mathbf{e}_s$, and passes through the planet origin. The spacecraft position can be projected onto this plane:

$$\mathbf{X}_{SNP} = \mathbf{X}_{ECS} - (\mathbf{e}_s \cdot \mathbf{X}_{ECS}) \mathbf{e}_s$$

When $\|\mathbf{X}_{SNP}\| < R_{\text{planet}}$ and if the spacecraft is in front of the solar normal plane $\mathbf{e}_s \cdot \mathbf{X}_{ECS} > 0$, the spacecraft is in eclipse.

Orienting the Spacecraft

The spacecraft is oriented by picking axes in the geometry of the spacecraft in the [Spacecraft Axes](#) feature, and then specifying what these axes should be oriented toward in the [Spacecraft Orientation](#) feature. The spacecraft can rotate about one, two, or all three axes and there can be parts of the spacecraft that articulate. Articulation is modeled via the Moving Mesh interface.

The primary and secondary axis specified by the user in the [Spacecraft Axes](#) feature are $\mathbf{e}_{SPA}$ and $\mathbf{e}_{SSA}$, respectively. They might not be orthogonal so a third vector is defined as $\mathbf{e}_{STA} = \mathbf{e}_{SPA} \times \mathbf{e}_{SSA} / \|\mathbf{e}_{SPA} \times \mathbf{e}_{SSA}\|$. From that, the secondary axis is redefined to form an orthonormal basis: $\mathbf{e}_{SSA} = \mathbf{e}_{STA} \times \mathbf{e}_{SPA}$. These three vectors form the columns of the orthonormal matrix that transforms from the spacecraft coordinate system (SCS) to the material coordinate system:

$$T_{MS} = \begin{bmatrix} \mathbf{e}_{SPA} & \mathbf{e}_{SSA} & \mathbf{e}_{STA} \end{bmatrix}$$

The primary and secondary orientation specified by the user in the [Spacecraft Orientation](#) feature are $\mathbf{e}_{P,t}$ and $\mathbf{e}_{S,t}$, respectively. They form a set of temporary vectors that might not be orthogonal so a third temporary vector is defined as $\mathbf{e}_{T,t} = (\mathbf{e}_{P,t} \times \mathbf{e}_{S,t}) / \|\mathbf{e}_{P,t} \times \mathbf{e}_{S,t}\|$. From that, the temporary secondary orientation is redefined to form an orthonormal basis: $\mathbf{e}_{S,t} = \mathbf{e}_{T,t} \times \mathbf{e}_{P,t}$.

These vectors are referred to as temporary vectors, because the spacecraft may be rotating about these three, so we have to define a set of three consecutive rotations about them, using the Rodrigues' rotation formula. Given a set of rotation angles, ω_p , ω_s , ω_t , about the temporary primary, secondary, and third orientation vectors of the SCS, the first rotation gives the vectors:

$$\begin{aligned} \mathbf{e}_{P,1} &= \mathbf{e}_{P,t} \cos \omega_p + (\mathbf{e}_{P,t} \times \mathbf{e}_{P,t}) \sin \omega_p + \mathbf{e}_{P,t} (\mathbf{e}_{P,t} \cdot \mathbf{e}_{P,t}) (1 - \cos(\omega_p)) \\ \mathbf{e}_{S,1} &= \mathbf{e}_{S,t} \cos \omega_p + (\mathbf{e}_{P,t} \times \mathbf{e}_{S,t}) \sin \omega_p + \mathbf{e}_{P,t} (\mathbf{e}_{P,t} \cdot \mathbf{e}_{S,t}) (1 - \cos(\omega_p)) \\ \mathbf{e}_{T,1} &= \mathbf{e}_{T,t} \cos \omega_p + (\mathbf{e}_{P,t} \times \mathbf{e}_{T,t}) \sin \omega_p + \mathbf{e}_{P,t} (\mathbf{e}_{P,t} \cdot \mathbf{e}_{T,t}) (1 - \cos(\omega_p)) \end{aligned}$$

The second rotation is about $\mathbf{e}_{S,t}$:

$$\begin{aligned} \mathbf{e}_{P,2} &= \mathbf{e}_{P,1} \cos \omega_s + (\mathbf{e}_{S,t} \times \mathbf{e}_{P,1}) \sin \omega_s + \mathbf{e}_{S,t} (\mathbf{e}_{S,t} \cdot \mathbf{e}_{P,1}) (1 - \cos(\omega_s)) \\ \mathbf{e}_{S,2} &= \mathbf{e}_{S,1} \cos \omega_s + (\mathbf{e}_{S,t} \times \mathbf{e}_{S,1}) \sin \omega_s + \mathbf{e}_{S,t} (\mathbf{e}_{S,t} \cdot \mathbf{e}_{S,1}) (1 - \cos(\omega_s)) \\ \mathbf{e}_{T,2} &= \mathbf{e}_{T,1} \cos \omega_s + (\mathbf{e}_{S,t} \times \mathbf{e}_{T,1}) \sin \omega_s + \mathbf{e}_{S,t} (\mathbf{e}_{S,t} \cdot \mathbf{e}_{T,1}) (1 - \cos(\omega_s)) \end{aligned}$$

The third rotation about $\mathbf{e}_{T,t}$ gives the primary, secondary, and third orientation vectors:

$$\begin{aligned} \mathbf{e}_{SPO} &= \mathbf{e}_{P,2} \cos \omega_t + (\mathbf{e}_{T,t} \times \mathbf{e}_{P,2}) \sin \omega_t + \mathbf{e}_{T,t} (\mathbf{e}_{T,t} \cdot \mathbf{e}_{P,2}) (1 - \cos(\omega_t)) \\ \mathbf{e}_{SSO} &= \mathbf{e}_{S,2} \cos \omega_t + (\mathbf{e}_{T,t} \times \mathbf{e}_{S,2}) \sin \omega_t + \mathbf{e}_{T,t} (\mathbf{e}_{T,t} \cdot \mathbf{e}_{S,2}) (1 - \cos(\omega_t)) \\ \mathbf{e}_{STO} &= \mathbf{e}_{T,2} \cos \omega_t + (\mathbf{e}_{T,t} \times \mathbf{e}_{T,2}) \sin \omega_t + \mathbf{e}_{T,t} (\mathbf{e}_{T,t} \cdot \mathbf{e}_{T,2}) (1 - \cos(\omega_t)) \end{aligned}$$

These vectors form the rows of an orthonormal matrix that transforms from ECS to the spacecraft coordinate system (SCS):

$$T_{SE} = \begin{bmatrix} \mathbf{e}_{SPO}^T \\ \mathbf{e}_{SSO}^T \\ \mathbf{e}_{STO}^T \end{bmatrix}$$

We now have the transformation matrix from ECS to the material frame:

$$T_{ME} = T_{MS}T_{SE}$$

This transformation matrix is used to transform all vectors in the ECS and the sources of radiative loads to the material frame.

Modeling the Planet

The planet accounts for two source of heat, the diffusely emitted infrared (IR) flux due to the nonzero temperature of the planet, and the diffusely reflected IR, or albedo, from the Sun.

The planet is approximated via a set of radiation sources on a spherical cap representing the part of the planet that can be seen from the spacecraft. The viewable planet can reasonably be tessellated into patches where the flux from each patch is reduced to a single point emitter, placed at the center of that patch. The magnitude of the source heat rate from that point emitter is the equivalent area of the patch, times the known flux and multiplied by a factor to account for the included angle of the patch.

The discretization of planet into patches is designed such that each source has an equal view factor to the spacecraft.

Since the total view factor from the relatively small spacecraft to the visible section of the planet is known, we know what the view factor should be for every point in the tessellation. This starts with one point immediately below the spacecraft, and the other

points arranged in concentric rings, as illustrated in [Figure 4-25](#). The number of ring and points per ring can be modified in the [Planet Properties](#) feature.

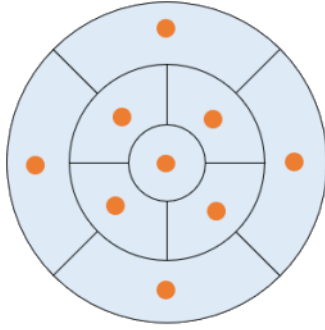


Figure 4-25: As viewed along the nadir direction, the arrangement of the radiation sources on the spherical cap representing the planet with 2 rings and 4 points per ring.

Modeling of Fast Tumbling

Fast tumbling is when the spacecraft is rotating about all of its axes sufficiently rapidly (as can happen during preoperational stages) such that the heat loads can be averaged over all surfaces. In this case, the heat loads applied to the surfaces is approximated to:

$$q = \frac{F_{\text{amb}}}{4} \left(q_{\text{solar}} + \sum_{p=1}^n q_p \right)$$

where F_{amb} is the view factor to ambient of all spacecraft surfaces, q_{solar} is the solar flux defined in the [Sun Properties](#) feature and q_p is the sum of the planet IR flux and albedo on each radiation source over the spherical cap as defined in [Modeling the Planet](#). n is the total number of radiation sources.

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
- Discrete Ordinates Method Implementation in 2D Axisymmetry
- 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

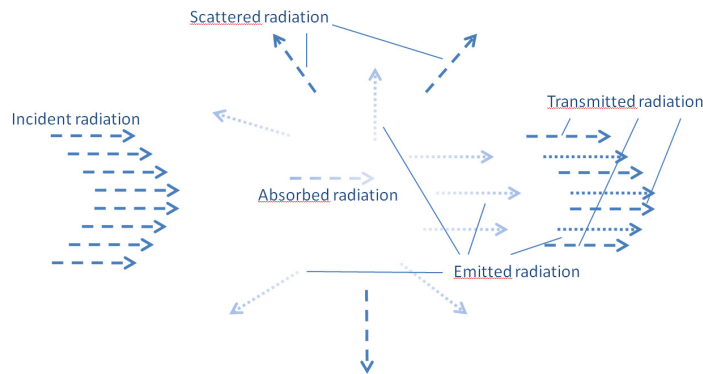


Figure 4-26: Example of interactions between participating media and radiation.

In some applications the medium is not completely transparent and the radiation rays interact with the medium.

Let $I(\Omega)$ denote the radiative intensity traveling in a given direction, Ω . Different kinds of interactions are observed:

- Absorption: The medium absorbs a fraction of the incident radiation. The amount of absorbed radiation is $\kappa I(\Omega)$, where κ is the absorption coefficient.
- Emission: The medium emits radiation in all directions. The amount of emitted radiative intensity is equal to κI_b , where I_b is the *blackbody radiation intensity*.
- Scattering: Part of the radiation coming from a given direction is scattered in other directions. The scattering properties of the medium are described by the scattering phase function $\phi(\Omega', \Omega)$, which gives the probability that a ray coming from one direction Ω' is scattered into the direction Ω . The phase function $\phi(\Omega', \Omega)$ satisfies:

$$\frac{1}{4\pi} \int_{4\pi} \phi(\Omega', \Omega) d\Omega' = 1$$

Radiative intensity in a given direction is attenuated and augmented by scattering:

- It is attenuated because a part of the incident radiation in this direction is scattered into other directions. The amount of radiation attenuated by scattering is $\sigma_s I(\Omega)$.
- It is augmented because a part of the radiative intensity coming from other directions is scattered in all direction, including the direction we are looking at. The amount of radiation augmented by scattering is obtained by integrating scattering coming from all directions Ω' :

$$\frac{\sigma_s}{4\pi} \int_{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega'$$

Radiative Transfer Equation

The balance of the radiative intensity including all contributions (propagation, emission, absorption, and scattering) can now be formulated. The general radiative transfer equation can be written as (see [Ref. 23](#)):

$$\Omega \cdot \nabla I(\Omega) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega' \quad (4-124)$$

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

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi} \quad (4-125)$$



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

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

where

$$I_b(T) = \frac{n_r^2 \sigma T^4}{\pi} \quad (4-126)$$

- Equation 4-126 is the blackbody radiation intensity and n_r is the refractive index
- ε is the surface emissivity, which is in the range $[0, 1]$
- $1 - \varepsilon$ is the diffusive reflectivity
- $\mathbf{n}$ is the outward normal vector
- $q_{r, \text{out}}$ 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 $\varepsilon = 1$. Thus $I(\Omega) = I_b(T)$.

The net heat flux corresponding to the balance between the energy received by the surface and emitted by the surface is defined by

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

where

$$q_{r, \text{in}} = - \int_{\mathbf{n} \cdot \Omega < 0} I(\Omega)(\mathbf{n} \cdot \Omega) d\Omega$$

This net heat flux accounts for the radiation coming from the domain where the RTE is solved for, the radiation coming from the exterior is not modeled and hence not accounted for.

If the surface is semitransparent, the radiative intensity transmitted diffusively or specularly. For an external boundary the external intensity, I_{ext} , is transmitted in the domain. The radiative intensity entering participating media along the Ω direction is

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

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

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

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. 23](#)):

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

In order to couple radiation in participating media, radiative heat flux is taken into account in addition to conductive heat flux. Recalling [Equation 4-18](#), the heat transfer equation reads:

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

and is implemented using following form:

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



The *discrete ordinates method* is implemented for 3D, 2D and 2D axisymmetric geometries.

Radiative intensity is defined for any direction Ω , because the angular space is continuous. In order to handle the radiative intensity equation numerically, the angular space is discretized.

The discrete ordinates method provides a discretization of angular space. The quadrature sets Level Symmetric Even, Level Symmetric Hybrid and Equal Weight Odd are designed using moment-matching conditions (see Ref. 24) and yield S_N approximations with $N(N+2)$ directions in 3D or $N(N+2)/2$ in 2D and 2D axisymmetric (see Ref. 23). The quasi-uniform quadrature set yields T_N approximations with $8N^2$ directions in 3D or $4N^2$ in 2D and 2D axisymmetric (Ref. 46, Ref. 47). These approximations are not designed using moment-matching conditions but allow for higher-order discretization compared with S_N approximations.

Thanks to angular space discretization, integrals over directions are replaced by numerical quadratures of discrete directions:

$$\int_{4\pi} I(\Omega) d\Omega \approx \sum_{j=1}^n w_j I_j$$

Depending on the value of N , a set of n dependent variables has to be defined and solved for $I_1, I_2, \dots, I_n$.

RADIATION IN PARTICIPATING MEDIA

Each dependent variable satisfies, in a cartesian coordinates system, 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 $I_1, I_2, \dots, I_n$:

$$I_{i, \text{bnd}} = \varepsilon I_{\text{b}}(T) + \frac{1 - \varepsilon}{\pi} q_{\text{r, out}} \quad \text{for all } \mathbf{S}_i \text{ such that } \mathbf{n} \cdot \mathbf{S}_i < 0 \quad (4-127)$$

with

$$q_{\text{r, out}} = \sum_{\mathbf{n} \cdot \mathbf{S}_j > 0} w_j I_j(\mathbf{n} \cdot \mathbf{S}_j)$$

In [Equation 4-127](#), 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 \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) defines the following boundary condition for radiative intensities $I_1, I_2, \dots, I_n$:

$$I_{i, \text{bnd}} = \varepsilon I_{\text{b}}(T) + \frac{\rho_{\text{d}}}{\pi} q_{\text{r, out}} + \frac{\tau_{\text{d}}}{\pi} q_{\text{r, out, ext}} + \tau_{\text{s}} I_{i, \text{ext}} \quad (4-128)$$

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

$$q_{\text{r, out}} = \sum_{\mathbf{n} \cdot \mathbf{S}_j > 0} w_j I_j(\mathbf{n} \cdot \mathbf{S}_j)$$

and

$$q_{\text{r, out, ext}} = - \sum_{\mathbf{n} \cdot \mathbf{S}_j \leq 0} w_j I_{j, \text{ext}}(\mathbf{n} \cdot \mathbf{S}_j)$$

In [Equation 4-128](#), 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

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

Assuming that a model is invariant in the z direction, and symmetric with respect to the xy -plane, then $I_{i^-} = I_{i^+}$. The radiative transfer equation can thus be solved for only half of the 3D directions for the discrete ordinates methods (DOM). For the $\mathbf{S}_{i^+}$ directions, the equation reads:

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

Note that the sum in the scattering term still contains all directions $\mathbf{S}_j$, not only the ones with a positive z component. Using $I_{i^-} = I_{i^+}$ and since quadrature sets are built symmetric ($w_{i^-} = w_{i^+}$), the scattering term is then recast

$$\mathbf{S}_{i^+} \cdot \nabla I_{i^+} = \kappa I_b(T) - \beta I_{i^+} + \frac{\sigma_s}{4\pi} \sum_{j=1}^{n/2} 2w_{j^+} I_{j^+} \frac{\phi(\mathbf{S}_{j^+}, \mathbf{S}_{i^+}) + \phi(\mathbf{S}_{j^-}, \mathbf{S}_{i^+})}{2}$$

The equation is now written only as a function of the radiative intensities associated to a positive z component direction, and only in the phase function do the negative

z component directions appear. The weights in 2D are redefined as the double of the 3D weights $\tilde{w}_i = 2w_i$, such that, in the end, the 2D formulation of the DOM is:

$$\mathbf{S}_{i^*} \cdot \nabla I_{i^*} = \kappa I_b(T) - \beta I_{i^*} + \frac{\sigma_s}{4\pi} \sum_{j=1}^{n/2} \tilde{w}_j I_{j^*} \frac{\phi(\mathbf{S}_j, \mathbf{S}_{i^*}) + \phi(\mathbf{S}_{j^*}, \mathbf{S}_{i^*})}{2} \quad (4-129)$$

In COMSOL Multiphysics, all phase functions $\phi(\mathbf{S}_j, \mathbf{S}_i)$ can be rewritten as functions of $\mathbf{S}_j \cdot \mathbf{S}_i$. For relatively simple phase functions, this allows for further simplification of the average between phase functions with positive and negative z component, which can lead to the equation being written without any negative z component directions. However, since more complex phase functions are also supported, this general form is preferred.

Also notice that

$$\int_{4\pi} I(\Omega) d\Omega \approx \sum_{j=1}^n w_j I_j = \sum_{j=1}^{n/2} w_j I_j + w_{j^*} I_{j^*} = \sum_{j=1}^{n/2} \tilde{w}_j I_{j^*} \quad (4-130)$$

Using results from [Equation 4-129](#) and [Equation 4-130](#) the DOM is formulated in 2D using only radiative intensities, I_{i^*} , on half of the 3D DOM directions, $\mathbf{S}_{i^*}$, except for the scattering term.

Discrete Ordinates Method Implementation in 2D Axisymmetry

Let's define a cylindrical coordinate system ($\mathbf{e}_r, \mathbf{e}_\varphi, \mathbf{e}_z$) and the direction Ω which coordinates (μ, η, ξ) in this coordinate system are expressed with its azimuthal angle ϕ with respect to the basis vector $\mathbf{e}_r$, and polar angle θ :

$$\Omega = \begin{bmatrix} \mu \\ \eta \\ \xi \end{bmatrix} = \begin{bmatrix} \sin(\theta) \cos(\phi) \\ \sin(\theta) \sin(\phi) \\ \cos(\theta) \end{bmatrix}$$

The radiative transfer equation has an additional term to account for the angular redistribution of the radiative intensity (see [Ref. 23](#)):

$$\nabla \cdot (\Omega I(\Omega)) - \frac{1}{r} \frac{\partial}{\partial \phi} (\eta I(\Omega)) = \kappa I_b(T) - \beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega'$$

For the discrete ordinates method, the angular redistribution term is approximated by a finite differences central scheme (see [Ref. 23](#)):

$$\frac{1}{r} \frac{\partial}{\partial \phi} (\eta_i I_i) = \frac{\alpha_{i+1/2} I_{i+1/2} - \alpha_{i-1/2} I_{i-1/2}}{r w_i}$$

In this equation, $I_{i-1/2}$ and $I_{i+1/2}$ are the intensities evaluated at the boundaries between the ordinates, and $\alpha_{i-1/2}$ and $\alpha_{i+1/2}$ are the angular redistribution coefficients. In turn, the intensities between ordinates are evaluated with a numerical scheme:

$$I_i = f I_{i+1/2} + (1-f) I_{i-1/2}$$

with the parameter $f = 0.5$, thus getting a diamond scheme, and the condition $I_{1/2} = I_1$. The angular redistribution coefficients are calculated from the quadrature set weights and directions:

$$\alpha_{i+1/2} - \alpha_{i-1/2} = \mu_i w_i$$

with the condition $\alpha_{1/2} = 0$.

The angular redistribution term is calculated with the equations above, and allows the discrete ordinates method to solve for 2D axisymmetric models. The handling of the other terms of the radiative transfer equation is similar to 2D, noting that the symmetry plane is the rz -plane. Therefore, as in 2D, only half of the 3D directions need to be solved for.



See [Discrete Ordinates Method Implementation in 2D](#) for more details.

P1 Approximation Theory

The **P1 approximation** is available as a radiation discretization method in [The Radiation in Participating Media Interface](#).

The P1 approximation is the simplest approximation provided by the method of spherical harmonics method (PN-method). This approximation provides additional accuracy compared to a Rosseland approximation even if it remains a very simple method. The P1 method relies on the following hypotheses:

- The media is optically thick media: $\tau \gg 1$, where τ is the optical thickness defined by the integral of absorption coefficient, κ , along a typical optical path:

$$\tau = \int_0^s \kappa ds$$

- The scattering is linear anisotropic.

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 anisotropic phase function.

The P1 approximation accounts for the radiation transfer equation

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

by solving following equation for $G = \int_{4\pi} I(\Omega) d\Omega$ (Ref. 23):

$$\nabla \cdot (-D_{P1} \nabla G) = -Q_r \quad (4-131)$$

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:

$$Q_r = \kappa(G - 4\pi I_b) \quad (4-132)$$

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. 23):

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

where $\mathbf{q}_{r, \text{net}}$ 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 4-131](#) is implemented.

In addition $\mathbf{Q}_r$, defined by [Equation 4-132](#), is added as an heat source in the heat transfer equation:

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

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

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

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

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

SEMITRANSSPARENT SURFACE

The [Semitransparent Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) boundary condition defines a boundary that emits radiation, and reflects one part of the incident intensity while the remaining is transmitted diffusively.

The net radiative heat flux, $\mathbf{q}_{r, \text{net}}$, 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:

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

RADIATIVE SOURCE

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

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

Radiation in Absorbing-Scattering Media Theory

The [Radiation in Absorbing-Scattering Media Interface](#) is available in 2D, 2D axisymmetric, and 3D components to model the propagation, attenuation, and scattering of an incident light within a semitransparent material considered to be nonemitting.

The radiative intensity $I(\Omega)$ at a given position following the Ω direction is the solution of the radiative transfer equation with no emission term (see [Ref. 23](#)):

$$\Omega \cdot \nabla I(\Omega) = -\beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega' \quad (4-133)$$

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 4-133](#), 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,

$$\nabla \cdot (-D_{\text{P1}} \nabla G) = -\kappa G, \quad (4-134)$$

with the following boundary condition for an opaque surface,

$$\mathbf{n} \cdot (-D_{\text{P1}} \nabla G) = \frac{\varepsilon}{2(2-\varepsilon)} G,$$

where D_{P1} is the P1 diffusion coefficient and ε the surface emissivity.

Polychromatic Radiation

As in the Surface-to-Surface Radiation Interface, polychromatic radiation is modeled using the stepwise-gray model ([Ref. 23](#)) to account for wavelength dependence of emission, absorption, and scattering through material properties; see the section [Wavelength Dependence of Surface Properties](#).

In the [Discrete Ordinates Method \(DOM\)](#), the Radiative Transfer Equation is solved for each spectral band:

$$\mathbf{S}_i \cdot \nabla I_{i,k} = \kappa_k I_{\text{b}}(T) \text{FEP}_k(T) - \beta_k I_{i,k} + \frac{\sigma_{\text{s},k}}{4\pi} \sum_{j=1}^N \omega_j I_{j,k} \phi_k(\mathbf{S}_j, \mathbf{S}_i),$$

where FEP is the fractional emissive power, also presented in the section [Wavelength Dependence of Surface Properties](#).

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

$$\nabla \cdot (-D_{\text{P1},k} \nabla G_k) = -Q_{\text{r},k}, \quad (4-135)$$

where

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

$$D_{P1,k} = \frac{1}{3\kappa_k + \sigma_{s,k}(3 - a_{1,k})}$$

- $a_{1,k}$ is the linear Legendre coefficient of the scattering phase function
- $Q_{r,k}$ is the radiative heat source:

$$Q_{r,k} = \kappa_k(G_k - 4\pi I_b \text{FEP}_k(T)). \quad (4-136)$$

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. 23](#)):

$$\mathbf{q}_{r,\lambda} = -\frac{4\pi}{3\beta_\lambda} \nabla I_{b,\lambda}$$

For a gray media it leads to

$$\mathbf{q}_r = -\frac{4\sigma}{3\beta_R} \nabla (n_r^2 T^4)$$

Assuming a constant refractive index, this can be rewritten as $\mathbf{q}_r = -k_R \nabla T$ with

$$k_R = \frac{16n_r^2 \sigma T^3}{3\beta_R}$$

and

$$\mathbf{q}_r = -\frac{16n_r^2 \sigma T^3}{3\beta_R} \nabla T$$

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

This section presents the moisture transport equations in various media solved in COMSOL Multiphysics, starting from mass conservation laws.

In this section:

- [Theory for Moisture Transport in Air](#)
- [Theory for Moisture Transport in Building Materials](#)

The moisture transport equations in porous media are detailed in [Theory for Moisture Transport in Porous Media](#), and moisture transport equations in solids are detailed in [Theory for Moisture Transport in Solids](#) in the *Porous Media Flow Module User's Guide*.

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

In the diluted species formulation, a transport equation is written in terms of vapor concentration. It accounts for convection, binary diffusion in air, and source terms. Writing it in a conservative form:

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

Since the molar mass is constant, it can be taken out of the derivatives. Then, to express the nonconservative form, the spatial derivative is distributed:

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

The continuity equation for moist air is:

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

With the diluted species 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, that is, the continuity equation reduces to:

$$\nabla \cdot \mathbf{u} = 0$$

In other terms, this formulation supposes that the flow of moist air is incompressible.

The incompressible hypothesis can be used to simplify the nonconservative form of the moisture transport equation. Additionally, in the diluted species formulation, the vapor diffusive flux is expressed as a function of the vapor concentration. [The Moisture Transport in Air Interface](#) solves for the following nonconservative equation, in which the moisture content variation is expressed through the transport of vapor concentration, c_v , by convection and binary diffusion in air:

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w = G \quad (4-137)$$

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

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

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

In the concentrated species formulation, a transport equation is written in terms of vapor mass fraction. It accounts for convection, binary diffusion in air, and source terms. Writing it in a conservative form:

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

The following relation between vapor concentration c_v and vapor mass fraction ω_v always stands:

$$M_v c_v = \rho_g \omega_v$$

where ρ_g is the moist air density (SI unit: kg/m^3). Therefore, we obtain the same equation in conservative form than the diluted species formulation:

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

The only difference is that the vapor diffusive flux $\mathbf{g}_w$ is written as a function of the vapor mass fraction ω_v instead of the vapor concentration c_v .

When the spatial and temporal variations of the vapor concentration induce significant variations of the moist air density, that is, when the moist air flow cannot be supposed incompressible, the velocity divergence term cannot be neglected. [The Moisture Transport in Air Interface](#) solves for the following nonconservative equation, in which the moisture content variation is expressed through the transport of vapor concentration, c_v , by convection, binary diffusion in air, and includes compressibility effects:

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w + M_v c_v \nabla \cdot \mathbf{u} = G \quad (4-138)$$

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

$$\omega_v = \frac{M_v c_v}{\rho_g}$$

By neglecting the spatial and temporal variations of ρ_g in the equations above, the equation of the diluted species formulation, [Equation 4-137](#), is obtained.

Theory for Moisture Transport in Building Materials

By considering that building materials are in general surrounded by an environment of constant atmospheric pressure and under a certain range of temperature and moisture conditions, specific equations are proposed in [Ref. 16](#) for moisture transport in this kind of materials.

Starting from the equation for the transport of moisture in porous media:

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

and using the diffusion model for the capillary flux:

$$\mathbf{g}_{lc} = -D_w \frac{\partial w(\phi_w)}{\partial \phi_w} \nabla \phi_w$$

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:

$$\frac{\partial w(\phi_w)}{\partial t} + \nabla \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = G$$

By introducing the moisture storage capacity, ξ , defined as

$$\xi = \frac{\partial w(\phi_w)}{\partial \phi_w}$$

the time-dependent term and the capillary flux can be rewritten so that the equation reads

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = G$$

$$\mathbf{g}_{lc} = -D_w \xi \nabla \phi_w$$

Finally, the diffusive flux for vapor can be written as

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

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 moisture transfer as the sum of the capillary moisture flux and the vapor diffusion flux ([Ref. 16](#)):

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = G \quad (4-139)$$

$$\mathbf{g}_{lc} = -D_w \frac{\partial w(\phi_w)}{\partial \phi_w} \nabla \phi_w = -D_w \xi \nabla \phi_w$$

$$\mathbf{g}_w = -\delta_p \nabla p_v(T) = -\delta_p \nabla (\phi_w p_{\text{sat}}(T))$$

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 tensor.
- 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.

See [Saturation State](#) for the definition of saturation pressure p_{sat} as a function of temperature.

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. At equilibrium, they are related through Kelvin's law, which is a relation of thermodynamical origin. As long as equilibrium is satisfied, the state variables are continuous between two media (at the interface between two porous media with different porosities, or between a porous and a free medium), and Kelvin's law is valid on both sides of the interface. In opposition, the liquid saturation or the moisture content depend on the porous matrix.

Note that capillary transport of liquid water should be considered in both regions.

Considering a porous medium made of a solid porous matrix of density ρ_s (SI unit: kg/m^3) which pores are filled with a fluid of density ρ_f (SI unit: kg/m^3). Noting its porosity ε_p (dimensionless), the density of the whole medium is

$$\varepsilon_p \rho_f + (1 - \varepsilon_p) \rho_s$$

As mentioned above, the fluid itself is made of liquid phase, and a gas phase. The liquid saturation s_l (dimensionless), defines the proportion of the pores that is filled with liquid. The fluid density can thus be expressed in terms of liquid density ρ_l and gas density ρ_g :

$$\rho_f = s_l \rho_l + (1 - s_l) \rho_g$$

The liquid phase is pure water, however, the gas phase is moist air, that is a mixture of dry air and water vapor. Thus, in turn, its density can be written in terms of dry air density ρ_a and water vapor density ρ_v with the help of the vapor mass fraction ω_v (dimensionless):

$$\rho_g = \rho_v + \rho_a$$

With $\rho_v = \omega_v \rho_g$ and $\rho_a = (1 - \omega_v) \rho_g$.

The moisture transport equations in porous media ensures the conservation of water, that is the total moisture content:

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

The mass conservation equation for the liquid water and water vapor account for convection, diffusion, evaporation or condensation, that is an exchange of mass between both phases, and possible mass sources for the vapor G_v (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$) and liquid G_l (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$) phases:

$$\frac{\partial}{\partial t}(\epsilon_p s_l \rho_l) + \nabla \cdot (\epsilon_p s_l \rho_l \tilde{\mathbf{u}}_l) + \nabla \cdot (\epsilon_p s_l \mathbf{j}_l) = -G_{\text{evap}} + G_l$$

$$\frac{\partial}{\partial t}(\epsilon_p (1-s_l) \omega_v \rho_g) + \nabla \cdot (\epsilon_p (1-s_l) \omega_v \rho_g \tilde{\mathbf{u}}_g) + \nabla \cdot (\epsilon_p (1-s_l) \mathbf{j}_v) = G_{\text{evap}} + G_v$$

Several notations are introduced:

- $\mathbf{g}_{lc}$ (SI unit: $\text{kg}/(\text{m}^2 \cdot \text{s})$) is the liquid water capillary flux, equal to $\epsilon_p s_l \mathbf{j}_l$.
- $\mathbf{g}_w$ (SI unit: $\text{kg}/(\text{m}^2 \cdot \text{s})$) is the moisture diffusive flux, equal to $\epsilon_p (1-s_l) \mathbf{j}_v$.
- $\mathbf{u}_l$ (SI unit: m/s) is the liquid water velocity, equal to $\epsilon_p s_l \tilde{\mathbf{u}}_l$, that should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area.
- $\mathbf{u}_v$ (SI unit: m/s) is the moist air velocity, in the sense of the Darcy velocity, equal to $\epsilon_p (1-s_l) \tilde{\mathbf{u}}_g$.

The water mass conservation equations can be written:

$$\frac{\partial}{\partial t}(\epsilon_p s_l \rho_l) + \nabla \cdot (\rho_l \mathbf{u}_l) + \nabla \cdot \mathbf{g}_{lc} = -G_{\text{evap}} + G_l \quad (4-140)$$

$$\frac{\partial}{\partial t}(\epsilon_p (1-s_l) \omega_v \rho_g) + \nabla \cdot (\omega_v \rho_g \mathbf{u}_g) + \nabla \cdot \mathbf{g}_w = G_{\text{evap}} + G_v \quad (4-141)$$

Different strategies exist to solve for these equations. For example, one can assume that, in the porous medium, the liquid and vapor phases are in equilibrium. In that case since the amount of liquid and vapor are linked through the sorption isotherm relation, one can solve for a unique equation for the total water content. The theory for this hypothesis is detailed in the [Equilibrium Moisture Transport](#) section below.

Otherwise, the [Nonequilibrium Moisture Transport](#) section describes the theory for modeling moisture transport in porous media with a vapor mass fraction and a liquid water saturation that do not verify the equilibrium conditions.

Equilibrium Moisture Transport

Assuming equilibrium, the total moisture content is defined by a sorption curve as

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

Summing [Equation 4-140](#) and [Equation 4-141](#), we obtain a single equation for the total moisture content, where the evaporation terms cancel each other out. The liquid and vapor sources are summed into a single source term G :

$$\frac{\partial}{\partial t}(w(\phi_w)) + \nabla \cdot (\omega_v \rho_g \mathbf{u}_g) + \nabla \cdot \mathbf{g}_w + \nabla \cdot (\rho_l \mathbf{u}_l) + \nabla \cdot \mathbf{g}_{lc} = G \quad (4-142)$$

Under the equilibrium hypothesis, the source term is distributed between the liquid and vapor phases proportionally to the amount of liquid and vapor respectively, that is:

$$G = G_v + G_l$$

$$G_v = (1 - s_l)G$$

$$G_l = s_l G$$

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/m^3) is the liquid water density, defined as a function of the temperature.
- $\mathbf{g}_{lc}$ is the liquid water capillary flux, defined either from the capillary pressure gradient by a Darcy's law:

$$\mathbf{g}_{lc} = \rho_l \frac{\kappa_{rl} \kappa}{\mu_l} \nabla p_c$$

or by a diffusion equation using the relative humidity:

$$\mathbf{g}_{lc} = -D_w \frac{\partial w(\phi_w)}{\partial \phi_w} \nabla \phi_w$$

- D_w (SI unit: m^2/s) is the moisture diffusivity.
- p_c is the capillary pressure, that can be related to the relative humidity, in the scope of the equilibrium hypothesis, using Kelvin's law:

$$p_c = -\frac{RT\rho_l}{M_v} \ln(\phi_w)$$

- G (SI unit: $kg/(m^3 \cdot s)$) is a moisture source (or sink). See the [Moisture Source](#) node.

The moisture source due to evaporation may be obtained from [Equation 4-141](#):

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

This moisture source is added as a mass source in the fluid flow equations solved for the computation of the moist air velocity field, $\mathbf{u}_g$.

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.

When equilibrium between liquid and vapor phases cannot be assumed, the liquid saturation and vapor mass fraction are not related, therefore, both the water conservation equations for the liquid phase and vapor phase must be solved separately.

$$\frac{\partial}{\partial t}(\epsilon_p s_l \rho_l) + \nabla \cdot (\rho_l \mathbf{u}_l) + \nabla \cdot \mathbf{g}_{lc} = -G_{\text{evap}} + G_l$$

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

Unlike the equilibrium assumption, the evaporation source is taken into account for the mass conservation equations, and not only for the heat source it generates in the heat transfer equation. Just like the liquid saturation and vapor mass fraction are not related, so are the mass sources G_l and G_v .

The following material properties, fields, and source are defined:

- 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/m^3) is the liquid water density, defined as a function of the temperature.
- $\mathbf{g}_{lc}$ is the liquid water capillary flux, defined either from the capillary pressure gradient by a Darcy's law:

$$\mathbf{g}_{lc} = \rho_l \frac{\kappa_{rl}\kappa}{\mu_l} \nabla p_c$$

or by a diffusion equation using the relative humidity:

$$\mathbf{g}_{lc} = -D_w \frac{\partial w(\phi_w)}{\partial \phi_w} \nabla \phi_w$$

- D_w (SI unit: m^2/s) is the moisture diffusivity.
- p_c (SI unit: Pa) is the capillary pressure, that can be related to the liquid saturation through different models, see [Capillary Pressure Models](#).
- G (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$) is a moisture source (or sink). See the [Moisture Source](#) node.

The moisture source due to evaporation is calculated as a deviation from the equilibrium state:

$$G_{\text{evap}} = K_{\text{evap}} \frac{M_v}{RT} (p_{v,\text{eq}} - p_v)$$

where K_{evap} is the evaporation rate (SI unit: $1/\text{s}$) which gives the speed with which the system comes back to equilibrium after a disturbance, R (SI unit: $\text{J}/(\text{mol} \cdot \text{K})$) is the universal gas constant, $p_{v,\text{eq}}$ (SI unit: Pa) is the equilibrium vapor pressure, and p_v (SI unit: Pa) is the partial pressure of water vapor.

In turn, the equilibrium vapor pressure can be given as a function of water activity a_w (dimensionless) and vapor saturation pressure p_{sat} (SI unit: Pa):

$$p_{v,\text{eq}} = a_w p_{\text{sat}}$$

The evaporation source is added as a mass source in the fluid flow equations solved for the computation of the moist air velocity field, $\mathbf{u}_g$.

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 liquid water saturation, and vapor mass fraction do not change with time and the first term in both moisture transport equations disappears.

Capillary Pressure Models

The capillary pressure can be either a user defined expression, or given by the van Genuchten or Brooks and Corey models.

VAN GENUCHTEN MODEL

The van Genuchten ([Ref. 49](#)) model gives the capillary pressure p_c (SI unit: Pa), as well as the liquid water relative permeability κ_{rl} (dimensionless), and the vapor relative permeability κ_{rv} (dimensionless). These quantities are expressed as functions of the liquid water saturation s_l (dimensionless), the entry capillary pressure p_{ec} (SI unit: Pa), and two dimensionless constitutive constants m_{vG} and l_{vG} .

$$p_c = p_{ec} \left(s_l^{-1/m_{vG}} - 1 \right)^{1-m_{vG}}$$

$$\kappa_{rl} = s_l^{l_{vG}} \left[1 - \left(1 - s_l^{1/m_{vG}} \right)^{m_{vG}} \right]^2$$

$$\kappa_{rg} = (1 - s_l)^{l_{vG}} \left(1 - s_l^{1/m_{vG}} \right)^{2m_{vG}}$$

The liquid water relative permeability is used in the computation of the liquid water capillary flux, while the vapor relative permeability is used to account for the presence of liquid water in the pores for the calculation of the moist air Darcy velocity. See the [Moisture Flow](#) multiphysics coupling.

BROOKS AND COREY MODEL

The Brooks and Corey ([Ref. 50](#)) model gives the capillary pressure p_c (SI unit: Pa), as well as the liquid water relative permeability κ_{rl} (dimensionless), and the vapor relative

permeability κ_{rv} (dimensionless). These quantities are expressed as functions of the liquid water saturation s_l (dimensionless), the entry capillary pressure p_{ec} (SI unit: Pa), and the pore distribution index λ_p (dimensionless).

$$p_c = p_{ec} s_l^{-1/\lambda_p}$$

$$\kappa_{rl} = s_l^{3+2/\lambda_p}$$

$$\kappa_{rg} = (1 - s_l)^2 \left(1 - s_l^{1+2/\lambda_p} \right)$$

The liquid water relative permeability is used in the computation of the liquid water capillary flux, while the vapor relative permeability is used to account for the presence of liquid water in the pores for the calculation of the moist air Darcy velocity. See the [Moisture Flow](#) multiphysics coupling.

Theory for the Heat Transfer Multiphysics Couplings

In this section:

- [Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces](#)
- [Theory for the Moisture Flow Interface](#)
- [Theory for the Thermoelectric Effect Interface](#)
- [Theory for the Building Materials Version of the Heat and Moisture Transport Interface](#)
- [Theory for the Moist Air Version of the Heat and Moisture Transport Interface](#)
- [Theory for the Moist Porous Media Version of the Heat and Moisture Transport Interface](#)
- [Theory for the Heat and Moisture Flow Interfaces](#)
- [Theory for the Electromagnetic Heating Interfaces](#)
- [Theory for the Thermal Stress Interface](#)

Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces

The following points of the theory of Nonisothermal Flow and Conjugate Heat Transfer are discussed in this part:

- [The Nonisothermal Flow and Conjugate Heat Transfer Equations](#)
- [Turbulent Nonisothermal Flow Theory](#)
- [Theory for the Nonisothermal Screen Boundary Condition](#)
- [Theory for the Interior Fan Boundary Condition](#)

See [Theory for the Single-Phase Flow Interfaces](#) and [Theory for the Turbulent Flow Interfaces](#) in the *CFD Module User's Guide* for a description of the theory related to laminar and turbulent single-phase flow interfaces.

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:

$$\begin{aligned}\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) &= 0 \\ \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} &= -\nabla p + \nabla \cdot \boldsymbol{\tau} + \mathbf{F}\end{aligned}\tag{4-143}$$

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

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

are not included by default because they are usually negligible. These terms can, however, be added by selecting corresponding checkboxes 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 checkbox.



- [The Heat Balance Equation](#)
- [Turbulent Nonisothermal Flow Theory](#)

TURBULENT NONISOTHERMAL FLOW THEORY

Turbulent energy transport is conceptually more complicated than energy transport in laminar flows because the turbulence is also a form of energy.

Equations for compressible turbulence are derived using the Favre average. The Favre average of a variable T is denoted $\tilde{T}$ and is defined by

$$\tilde{T} = \frac{\overline{\rho T}}{\bar{\rho}}$$

where the bar denotes the usual Reynolds average. The full field is then decomposed as

$$T = \tilde{T} + T''$$

With this notation the energy balance equation becomes

$$\begin{aligned} \frac{\partial}{\partial t} \left(\bar{\rho} \left(\tilde{E} + \frac{\tilde{u}_i \tilde{u}_i}{2} \right) + \frac{\overline{\rho u_i'' u_i''}}{2} \right) + \frac{\partial}{\partial x_j} \left(\bar{\rho} \tilde{u}_j \left(\tilde{H} + \frac{\tilde{u}_i \tilde{u}_i}{2} \right) + \tilde{u}_j \frac{\overline{\rho u_i'' u_i''}}{2} \right) = \\ \frac{\partial}{\partial x_j} \left(-q_j - \overline{\rho u_j'' H''} + \tau_{ij} u_i'' - \frac{\overline{\rho u_j'' u_i'' u_i''}}{2} \right) + \frac{\partial}{\partial x_j} (\tilde{u}_i (\tau_{ij} - \overline{\rho u_i'' u_j''})) \end{aligned} \quad (4-144)$$

where H is the enthalpy. The vector

$$q_j = -\lambda \frac{\partial T}{\partial x_j} \quad (4-145)$$

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*:

$$-\overline{\rho u_i'' u_j''} = \bar{\rho} \tau_{ij}^T = \mu_T \left(\frac{\partial \tilde{u}_i}{\partial x_j} + \frac{\partial \tilde{u}_j}{\partial x_i} \right) - \frac{2}{3} \mu \frac{\partial \tilde{u}_k}{\partial x_k} \delta_{ij} - \frac{2}{3} \bar{\rho} k \delta_{ij} \quad (4-146)$$

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

$$\bar{\rho} k = \frac{1}{2} \overline{\rho u_i'' u_i''} \quad (4-147)$$

The correlation between u_j'' and H'' in Equation 4-144 is the turbulent transport of heat. It is modeled analogously to the laminar conductive heat flux

$$\overline{\rho u_j'' H''} = q_j^T = -\lambda_T \frac{\partial \tilde{T}}{\partial x_j} = -\frac{\mu_T C_p}{\text{Pr}_T} \frac{\partial \tilde{T}}{\partial x_j} \quad (4-148)$$

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

$$\overline{\tau_{ij}u_i''} - \frac{\overline{\rho u_j'' u_i'' u_i''}}{2} = \left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \quad (4-149)$$

Inserting Equation 4-145, Equation 4-146, Equation 4-147, Equation 4-148 and Equation 4-149 into Equation 4-144 gives

$$\begin{aligned} \frac{\partial}{\partial t} \left(\bar{\rho} \left(\tilde{E} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k \right) \right) + \frac{\partial}{\partial x_j} \left(\bar{\rho} \tilde{u}_j \left(\tilde{H} + \frac{\tilde{u}_i \tilde{u}_i}{2} + k \right) \right) = \\ \frac{\partial}{\partial x_j} \left(-q_j - q_j^T + \left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \frac{\partial}{\partial x_j} \left(\tilde{u}_i (\bar{\tau}_{ij} + \bar{\rho} \tau_{ij}^T) \right) \end{aligned} \quad (4-150)$$

The Favre average can also be applied to the momentum equation, which, using Equation 4-146, can be written

$$\frac{\partial}{\partial t} (\bar{\rho} \tilde{u}_i) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j \tilde{u}_i) = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j} (\bar{\tau}_{ij} + \bar{\rho} \tau_{ij}^T) \quad (4-151)$$

Taking the inner product between $\tilde{u}_i$ and Equation 4-151 results in an equation for the resolved kinetic energy, which can be subtracted from Equation 4-150 with the following result:

$$\begin{aligned} \frac{\partial}{\partial t} (\bar{\rho} (\tilde{E} + k)) + \frac{\partial}{\partial x_j} (\bar{\rho} \tilde{u}_j (\tilde{E} + k)) = \\ -\bar{p} \frac{\partial \tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j} \left(-q_j - q_j^T + \left(\mu + \frac{\mu_T}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right) + \frac{\partial}{\partial x_j} (\tilde{u}_i (\bar{\tau}_{ij} + \bar{\rho} \tau_{ij}^T)) \end{aligned} \quad (4-152)$$

where the relation

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

has been used.

According to Wilcox (Ref. 27), it is usually a good approximation to neglect the contributions of k for flows with Mach numbers up to the supersonic range. This gives the following approximation of Equation 4-152:

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E}) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial\tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}(-q_j - q_j^T) + \frac{\partial}{\partial x_j}(\tilde{u}_i(\bar{\tau}_{ij} + \bar{\rho}\tau_{ij}^T)) \quad (4-153)$$

Larsson (Ref. 28) 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

$$\frac{\partial}{\partial t}(\bar{\rho}\tilde{E}) + \frac{\partial}{\partial x_j}(\bar{\rho}\tilde{u}_j\tilde{E}) = -\bar{p}\frac{\partial\tilde{u}_j}{\partial x_j} + \frac{\partial}{\partial x_j}\left((\lambda + \lambda_T)\frac{\partial\tilde{T}}{\partial x_j}\right) + \frac{\partial}{\partial x_j}(\tilde{u}_i\tilde{\tau}_{ij}^{\text{tot}}) \quad (4-154)$$

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 4-154 is completely analogous to the laminar energy equation of Equation 4-15 and can be expanded using the same theory to get the temperature equation similar to Equation 4-18 (see for example Ref. 28):

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

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. 29](#), 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

$$Pr_T = \left(\frac{1}{2Pr_{T\infty}} + \frac{0.3C_p\mu_T}{\lambda\sqrt{Pr_{T\infty}}} \left(\frac{0.3C_p\mu_T}{\lambda} \right)^2 \left(1 - \exp\left(-\frac{\lambda}{0.3C_p\mu_T\sqrt{Pr_{T\infty}}} \right) \right) \right)^{-1} \quad (4-155)$$

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

Extended Kays–Crawford Weigand and others ([Ref. 30](#)) suggested an extension of [Equation 4-155](#) 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.

Standard The standard temperature wall function takes into account conduction in the boundary layer. 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. 31](#)):

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

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 Kármán constant equal to 0.41.

High viscous dissipation at wall Launder and Spalding propose a temperature wall function that accounts for conduction as well as viscous dissipation. The heat flux is written

$$q_{wf} = \begin{cases} \frac{\rho u_\tau}{\text{Pr} u^+} \left[C_p (T_w - T_f) - \frac{1}{2} \text{Pr} U^2 \right] & \text{for } \delta_w^+ < \delta_{wc}^+ \\ \frac{\rho u_\tau}{\text{Pr}_T (u^+ + P)} \left[C_p (T_w - T_f) - \frac{1}{2} (\text{Pr} U_c^2 + \text{Pr}_T (U^2 - U_c^2)) \right] & \text{for } \delta_w^+ \geq \delta_{wc}^+ \end{cases}$$

Variables with an index “c” are evaluated at the critical distance, which corresponds to the switch from the laminar sublayer to the logarithmic layer. The critical wall distance δ_{wc}^+ , which depends on the turbulent parameters κ and B , is estimated from a second-order accurate solution:

$$\delta_{wc}^+ = \frac{X_0}{\kappa} \left[2 - X_0 + \sqrt{(X_0 - 1)^2 + 2 \left(-\kappa_0 B_0 + \kappa B - \ln \left(\frac{\kappa}{\kappa_0} \right) \right)} \right]$$

with $\kappa_0 = 0.41$, $B_0 = 5.2$, and $X_0 = 4.53554291157957$.

The velocity at δ_{wc}^+ is then calculated from a velocity wall function, according to the turbulence model and wall treatment chosen to keep consistency. The velocity wall

function used for the determination of U_c when the L-VEL or a LES turbulence model is chosen is based on a blending of the linear and logarithmic laws. It gives very similar results to the wall function actually used for the calculation of the velocity at the first cell which is based on the wall Reynolds number.

P is a measure of the resistance of the boundary layer to heat transport, and is estimated with Jayatilleke's formula:

$$P = 9.21 \left[\left(\frac{Pr}{Pr_T} \right)^{3/4} - 1 \right] \left[1 + 0.28 \exp \left(-0.007 \frac{Pr}{Pr_T} \right) \right]$$

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:

$$[H_0]^+_- = 0 \quad (4-156)$$

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 the direction is along the 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 the 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](#)
- [Stefan Flow at Walls](#)
- [Turbulent Moisture Flow Theory](#)
- [Theory for the Screen Boundary Condition in Moisture Flow](#)
- [Theory for the Interior Fan Boundary Condition in Moisture Flow](#)

See [Theory for the Single-Phase Flow Interfaces](#) and [Theory for the Turbulent Flow Interfaces](#) in the *CFD Module User's Guide* for a description of the theory related to laminar and turbulent single-phase flow interfaces.

THE MOISTURE FLOW EQUATIONS

This module includes the Moisture Flow predefined multiphysics coupling to simulate systems in which the airflow depends on moisture content through its density and viscosity. It can be used in free and hygroscopic porous media containing moist air.

The density and viscosity of air might also vary due to temperature variations. See [Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces](#) for details on this other kind of dependency.

Moisture Flow in Moist Air Domains

In moist air domains, the Moisture Flow interface contains the fully compressible formulation of the continuity and momentum equations. For laminar flow they read:

$$\begin{aligned} \frac{\partial \rho_g}{\partial t} + \nabla \cdot (\rho_g \mathbf{u}) &= 0 \\ \rho_g \frac{\partial \mathbf{u}}{\partial t} + \rho_g \mathbf{u} \cdot \nabla \mathbf{u} &= -\nabla p + \nabla \cdot \boldsymbol{\tau} + \mathbf{F} \end{aligned} \tag{4-157}$$

where

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

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

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

It also solves the equation for moisture transport in air, given in [Equation 4-137](#) by

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

where

- M_v is the molar mass of water vapor (SI unit: kg/mol)
- c_v is the vapor concentration (SI unit: mol/m³)
- $\mathbf{g}_w$ is the moisture flux by diffusion (SI unit: kg/(m²·s)):

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

- D (SI unit: m²/s) is the vapor diffusion coefficient in air.
- G contains moisture sources (SI unit: kg/(m³·s))

When the spatial and temporal variations of the vapor concentration induce significant variations of the moist air density, the compressibility effects must be taken into account in the moisture transport equation:

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

with

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

where the vapor mass fraction, ω_v , is defined as:

$$\omega_v = \frac{M_v c_v}{\rho_g}$$

Moisture Flow in Hygroscopic Porous Medium domains

In porous medium domains, the Moisture Flow interface solves the Brinkman equations. They read:

$$\begin{aligned} \frac{1}{\varepsilon_p} \rho_g \frac{\partial \mathbf{u}}{\partial t} + \frac{1}{\varepsilon_p} \rho_g (\mathbf{u} \cdot \nabla) \mathbf{u} \frac{1}{\varepsilon_p} &= \nabla \cdot (-p \mathbf{I} + \mathbf{K}) - \left(\mu_g \kappa^{-1} + \beta \rho_g |\mathbf{u}| + \frac{Q_m}{\varepsilon_p^2} \right) \mathbf{u} + \mathbf{F} \\ \frac{\partial \varepsilon_p \rho_g}{\partial t} + \nabla \cdot (\rho_g \mathbf{u}) &= Q_m \end{aligned} \quad (4-158)$$

where:

- ε_p is the porosity (dimensionless), when accounting for the liquid water phase into the porous matrix. It is obtained by multiplying the porosity of the dry porous medium by $(1 - s_l)$, where s_l is the liquid water saturation.
- ρ_g is the moist air density (SI unit: kg/m³)
- $\mathbf{u}$ is the velocity vector (SI unit: m/s)
- p is the pressure (SI unit: Pa)
- $\mathbf{K}$ is the viscous stress tensor (SI unit: Pa), equal to:

$$\mathbf{K} = \mu_g \frac{1}{\varepsilon_p} (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu_g \frac{1}{\varepsilon_p} (\nabla \cdot \mathbf{u}) \mathbf{I}$$

- μ_g is the moist air dynamic viscosity (SI unit: Pa.s)
- κ is the permeability (SI unit: m²)
- Q_m is the mass source or sink in moist air, due to evaporation or condensation. It is defined as

$$Q_m = G_{\text{evap}}$$

Under the equilibrium hypothesis, G_{evap} is obtained from the vapor mass conservation equation:

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

When this hypothesis is not applicable, G_{evap} is defined as

$$G_{\text{evap}} = K_{\text{evap}} \frac{M_v}{RT} (p_{v, \text{eq}} - p_v)$$

See [Theory for Moisture Transport in Porous Media](#) for details.

- $\mathbf{F}$ is the body force vector (SI unit: N/m^3)

It also solves the equation for moisture transport in hygroscopic porous media, given in [Equation 4-142](#) for the equilibrium hypothesis, and in [Equation 4-140](#) and [Equation 4-141](#) for the nonequilibrium hypothesis.

STEFAN FLOW AT WALLS

Surface reactions like evaporation and condensation result in a net moisture flux $\mathbf{g}_w$ between the boundary and the domain. The resulting effective velocity at the boundary is the Stefan velocity u_{Stefan} , defined as

$$\rho_g u_{\text{Stefan}} = \mathbf{n} \cdot \mathbf{g}_w$$

The Moisture Flow interface allows to use the Stefan velocity as a leakage velocity at walls in the fluid flow equations.

TURBULENT MOISTURE FLOW THEORY

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

$$\tilde{c}_v = \frac{\overline{\rho_g c_v}}{\rho_g}$$

where the bar denotes the usual Reynolds average. The full field can be decomposed as the sum of the Favre average and the Favre fluctuation:

$$c_v = \tilde{c}_v + c_v''$$

By applying this decomposition and averaging the moisture transport equations, unclosed terms are introduced in the equations. Among them, the turbulent moisture transport flux contains the velocity and vapor concentration fluctuations:

$$u_j'' \frac{\partial c_v''}{\partial x_j}$$

This flux is modeled using a gradient based assumption, where the additional transport is related to the turbulent viscosity ν_T through the turbulent Schmidt number Sc_T . The turbulent diffusivity D_T is defined by

$$D_T = \frac{\nu_T}{Sc_T}$$

The following moisture transport equation is solved:

$$M_v \frac{\partial \tilde{c}_v}{\partial t} + M_v \tilde{u}_j \frac{\partial \tilde{c}_v}{\partial x_j} + M_v \frac{\partial}{\partial x_j} \left(-(D + D_T) \frac{\partial \tilde{c}_v}{\partial x_j} \right) = G \quad (4-159)$$

Kays–Crawford Model for Turbulent Diffusivity

By analogy with turbulent heat transfer, the turbulent Schmidt number Sc_T is given by (Ref. 29)

$$Sc_T = \left(\frac{1}{2Sc_{T\infty}} + \frac{0.3\mu_T Sc}{\mu \sqrt{Sc_{T\infty}}} - \left(0.3 \frac{\mu_T Sc}{\mu} \right)^2 \left(1 - \exp \left(-\frac{\mu}{0.3\mu_T Sc \sqrt{Sc_{T\infty}}} \right) \right) \right)^{-1} \quad (4-160)$$

where the Schmidt number at infinity is $Sc_{T\infty} = 0.85$ and the Schmidt number is defined as

$$Sc = \frac{\nu}{D}$$

where ν is the kinematic viscosity.

Moisture Transport Wall Functions

Analogous to the single-phase flow wall functions (see Wall Functions described for the Wall boundary condition), there is a theoretical gap between the solid wall and the computational domain for the fluid and relative humidity fields. This gap is often ignored when the computational geometry is drawn.

Assuming that the turbulent heat and moisture transfer in the near-wall region are analogous, the same type of wall functions used for the temperature (Ref. 31) is also applicable for the moisture transport. The moisture transfer wall function is formulated as a function of the turbulent Schmidt number, instead of the corresponding Prandtl number.

The moisture flux at the lift-off position between the air with vapor mass fraction $\omega_{v,a}$ and a wall with vapor mass fraction $\omega_{v,w}$, is:

$$g_{wf} = \frac{\rho_g u_\tau (\omega_{v,w} - \omega_{v,a})}{\omega_w^+}$$

where u_τ is the friction velocity, and ω_w^+ is the dimensionless mass fraction given by (Ref. 31):

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

where in turn

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

where κ is the von Kármán 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:

$$[c_v]_-^+ = 0 \quad (4-161)$$

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

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

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

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

Theory for the Thermoelectric Effect Interface

[The Thermoelectric Effect Interface](#) implements thermoelectric effect, which is the direct conversion of temperature differences to electric voltage or vice versa. Devices such as thermoelectric coolers for electronic cooling or portable refrigerators rely on this effect. While Joule heating (resistive heating) is an irreversible phenomenon, the thermoelectric effect is, in principle, reversible.

Historically, the thermoelectric effect is known by three different names, reflecting its discovery in experiments by Seebeck, Peltier, and Thomson. The *Seebeck effect* is the conversion of temperature differences into electricity, the *Peltier effect* is the conversion of electricity to temperature differences, and the *Thomson effect* is heat produced by the product of current density and temperature gradients. These effects are thermodynamically related by the Thomson relations:

$$P = ST$$

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

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

$$\mathbf{q} = -k\nabla T + P\mathbf{J} \quad (4-162)$$

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

$$\mathbf{J} = -\sigma(\nabla V + S\nabla T) \quad (4-163)$$

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 electric 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 electric current in an immobile solid read

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

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

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 4-162](#) and [Equation 4-163](#), 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 4-163](#) 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 4-162](#).

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 4-162](#). This time consider the term $-T \mathbf{J} \cdot \nabla S$. Assuming that S is function of T , then:

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

Theory for the Building Materials Version of the Heat and Moisture Transport Interface

The Building Materials version of the [Heat and Moisture](#) multiphysics coupling implements the following equations for heat and moisture transport, derived from [Ref. 16](#):

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + \nabla \cdot (-k_{\text{eff}} \nabla T - L_v \delta_p \nabla(\phi_w p_{\text{sat}})) = Q \quad (4-164)$$

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (-\xi D_w \nabla \phi_w - \delta_p \nabla(\phi_w p_{\text{sat}})) = G \quad (4-165)$$

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:

$$\rho_g C_p \frac{\partial T}{\partial t} + \rho_g C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (4-166)$$

$$\mathbf{q} = -k \nabla T \quad (4-167)$$

$$Q = -(C_{p,v} - C_{p,a}) \mathbf{g}_w \cdot \nabla T \quad (4-168)$$

$$M_v \frac{\partial c_v}{\partial t} + M_v \nabla \cdot (c_v \mathbf{u}) + \nabla \cdot \mathbf{g}_w = G \quad (4-169)$$

$$\mathbf{g}_w = -M_v D \nabla c_v \quad (4-170)$$

$$c_v = \phi_w c_{\text{sat}} \quad (4-171)$$

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:

$$q_{\text{evap}} = -L_v g_{\text{evap}} \quad (4-172)$$

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 4-168](#))
- The definition of the latent heat source from the evaporation source on surfaces ([Equation 4-172](#))
- The definition of the vapor saturation in function of the temperature ([Equation 4-171](#))

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:

$$(\rho C_p)_{\text{eff}} \frac{\partial T}{\partial t} + (\rho_g C_{p,g} \mathbf{u}_g + \rho_l C_{p,l} \mathbf{u}_l) \cdot \nabla T + \nabla \cdot \mathbf{q} = Q + Q_{\text{evap}} \quad (4-173)$$

$$\mathbf{q} = -k_{\text{eff}} \nabla T \quad (4-174)$$

$$Q = -[(C_{p,v} - C_{p,a}) \mathbf{g}_w + C_{p,l} \mathbf{g}_{lc}] \cdot \nabla T \quad (4-175)$$

$$Q_{\text{evap}} = -L_v G_{\text{evap}} \quad (4-176)$$

Under the equilibrium hypothesis, G_{evap} is obtained from the vapor mass conservation equation:

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

When this hypothesis is not applicable, G_{evap} is defined as

$$G_{\text{evap}} = K_{\text{evap}} \frac{M_v}{RT} (p_{v, \text{eq}} - p_v)$$

See [Theory for Moisture Transport in Porous Media](#) for details.

It also solves the equation for moisture transport in hygroscopic porous media, given in [Equation 4-142](#) for the equilibrium hypothesis, and in [Equation 4-140](#) and [Equation 4-141](#) for the nonequilibrium hypothesis.

See [Theory for Heat Transfer in Moist 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 define the convective heat flux
- The diffusive flux of enthalpy in moist air and the liquid capillary flux ([Equation 4-175](#))
- The definition of the latent heat source from the evaporation source in domains ([Equation 4-176](#))
- 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 the [Electromagnetic Heating](#) multiphysics coupling feature in common. It 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 4-27](#).

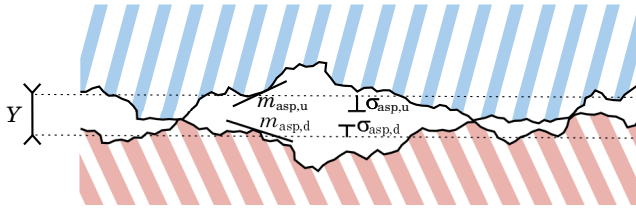


Figure 4-27: Contacting surfaces at the microscopic level.

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](#)):

$$\sigma_{asp} = \sqrt{\sigma_{u, asp}^2 + \sigma_{d, asp}^2} \quad m_{asp} = \sqrt{m_{u, asp}^2 + m_{d, asp}^2}$$

CONSTRICION CONDUCTANCE

Cooper–Mikic–Yovanovich (CMY) Correlation

The Cooper–Mikic–Yovanovich (CMY) correlation is valid for isotropic rough surfaces and has been formulated using a model assuming plastic deformation of the surface asperities. However, this model does not compute nor store the plastic deformations of the asperities. It means that, despite that a plastic deformation of the asperities is assumed, this contact model has no memory. For example, if a load is applied twice the thermal contact is identical in both cases. The Cooper–Mikic–Yovanovich (CMY) correlation relates h_c to the asperities and pressure load at the contact interface:

$$h_c = 1.25 k_{\text{contact}} \frac{m_{\text{asp}}}{\sigma_{\text{asp}}} \left(\frac{p}{H_c} \right)^{0.95}$$

Here, H_c is the microhardness of the softer material, p is the contact pressure, and k_{contact} is the harmonic mean of the contacting surface conductivities:

$$k_{\text{contact}} = \frac{2k_u k_d}{k_u + k_d}$$



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:

$$\frac{p}{H_c} = \left(\frac{p}{c_1 \left(1.62 \frac{\sigma_{\text{asp}}}{m_{\text{asp}} \sigma_0} \right)^{c_2}} \right)^{\frac{1}{(1 + 0.071 c_2)}}$$

The coefficients c_1 and c_2 are the Vickers correlation coefficient and size index, respectively, and σ_0 is equal to 1 μm . For materials with Brinell hardness between 1.30 and 7.60 GPa, c_1 and c_2 are given by the correlation below (4.16.1 in [Ref. 1](#)):

$$\frac{c_1}{H_0} = 4.0 - 5.77 \frac{H_B}{H_0} + 4.0 \left(\frac{H_B}{H_0} \right)^2 - 0.61 \left(\frac{H_B}{H_0} \right)^3$$

$$c_2 = -0.37 + 0.442 \frac{H_B}{c_1}$$

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 Young's moduli of the two contacting surfaces and ν_u and ν_d are 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 4-27), and M_g is the gas parameter equal to:

$$M_g = \alpha \beta \Lambda \quad \Lambda = \frac{k_B T_g}{\sqrt{2} \pi D_g^2 p_g}$$

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 4-28 shows examples of possible situations in which this type of geometry reduction can be applied.

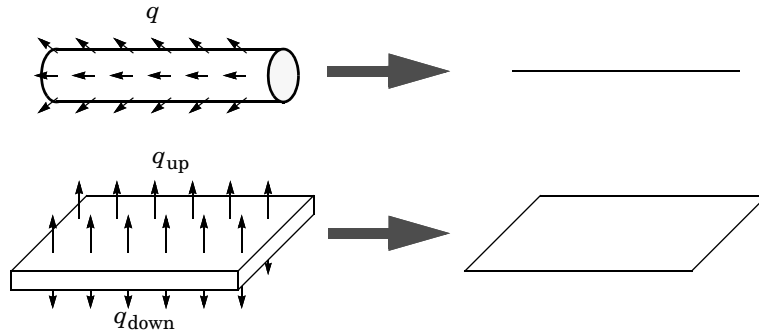


Figure 4-28: Geometry reduction from 3D to 1D (top) and from 3D to 2D (bottom).

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 4-28 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:

$$d_z \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = d_z Q + q_0 \quad (4-177)$$

$$d_z \rho C_p \frac{\partial T}{\partial t} + \rho C_p d_z \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = d_z Q + q_0 \quad (4-178)$$

Here d_z is the thickness of the domain in the out-of-plane direction. Here, the conductive heat flux, $\mathbf{q}$, becomes

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

1D AXISYMMETRIC GEOMETRIES

In 1D axisymmetric geometries, the temperature is assumed to be constant in the out-of-plane direction (z direction with default spatial coordinate names) in addition to the axisymmetry (ϕ -coordinate with default spatial coordinate names). The equation for heat transfer in solids, [Equation 6-15](#) is replaced by

$$(2\pi r d_z) \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = (2\pi r d_z) Q + q_0 \quad (4-179)$$

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

$$(2\pi r d_z) \rho C_p \frac{\partial T}{\partial t} + (2\pi r d_z) \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = (2\pi r d_z) Q + q_0 \quad (4-180)$$

Here, the conductive heat flux, $\mathbf{q}$, becomes

$$\mathbf{q} = -(2\pi r d_z) k \nabla T$$

1D GEOMETRIES

In 1D geometries, the temperature is assumed to be constant in the radial direction. The equation for heat transfer in solids, [Equation 6-15](#) is replaced by

$$A_c \rho C_p \frac{\partial T}{\partial t} + \nabla \cdot \mathbf{q} = A_c Q + q_0 \quad (4-181)$$

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

$$A_c \rho C_p \frac{\partial T}{\partial t} + A_c \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = A_c Q + q_0 \quad (4-182)$$

Here, the conductive heat flux, $\mathbf{q}$, becomes

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

Out-of-plane flux conditions would apply to the exterior boundaries of the domain if the 1D geometry was seen as a cylinder. With the geometry reduction process, this heat flux condition is mathematically expressed using the cross section perimeter, P_c , as in:

$$q_0 = P_c q_{0,z}$$

where $q_{0,z}$ is the heat flux density distributed along the cross section perimeter.

Convective Heat Transfer Correlations

One of the most common boundary conditions when modeling heat transfer is convective cooling or heating whereby a fluid cools or heats a surface by natural or forced convection. In principle, it is possible to model this process in two ways:

- Using a heat transfer coefficient on the surfaces
- Extending the model to describe the flow and heat transfer in the surrounding fluid

The second approach is the most accurate if the geometry or the external flow is complicated. The Heat Transfer Module includes the Conjugate Heat Transfer predefined multiphysics coupling and the CFD Module includes the Nonisothermal Flow predefined multiphysics coupling for this purpose. However, such a simulation can become costly, both in terms of computational time and memory requirement.

The first method is simple, yet powerful and efficient. The convective heat flux on the boundaries in contact with the fluid is then modeled as being proportional to the temperature difference across a fictitious thermal boundary layer. Mathematically, the heat flux is described by the equation

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\text{ext}} - T)$$

where h is a heat transfer coefficient and T_{ext} the temperature of the external fluid far from the boundary.

The main difficulty in using heat transfer coefficients is in calculating or specifying the appropriate value of the h coefficient. That coefficient depends on the fluid's material properties, and the surface temperature — and, for forced convection, also on the fluid's flow rate. In addition, the geometrical configuration affects the coefficient. The Heat Transfer interface has built-in functions for the heat transfer coefficients. For most engineering purposes, the use of such coefficients is an accurate and numerically efficient modeling approach.

In this section:

- [Defining the Heat Transfer Coefficients](#)
- [Nature of the Flow — The Grashof Number](#)
- [Heat Transfer Coefficients — External Natural Convection](#)

- Heat Transfer Coefficients — Internal Natural Convection
- Heat Transfer Coefficients — External Forced Convection
- Heat Transfer Coefficients — Internal Forced Convection
- Using the Heat and Mass Transfer Analogy for the Evaluation of Moisture Transfer Coefficients

Defining the Heat Transfer Coefficients

It is possible to divide the convective heat flux into four main categories depending on the type of convection condition (natural or forced) and on the type of geometry (internal or external flow). In addition, these cases can all experience either laminar or turbulent flow conditions, resulting in eight types of convection, as in Figure 4-29.

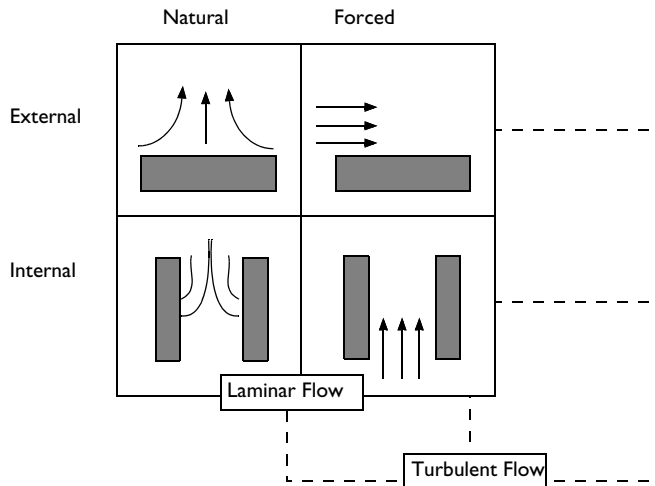


Figure 4-29: The eight categories of convective heat flux.

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·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 the 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 the 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

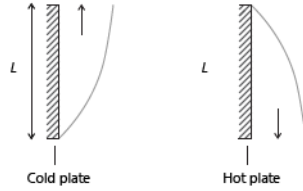


Figure 4-30: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a vertical wall.

The following correlations correspond to equations 9.26 and 9.27 in [Ref. 21](#):

$$h = \begin{cases} \frac{k}{L} \left(0.68 + \frac{0.67 \text{Ra}_L^{1/4}}{\left(1 + \left(\frac{0.492k}{\mu C_p} \right)^{9/16} \right)^{4/9}} \right) & \text{if } \text{Ra}_L \leq 10^9 \\ \frac{k}{L} \left(0.825 + \frac{0.387 \text{Ra}_L^{1/6}}{\left(1 + \left(\frac{0.492k}{\mu C_p} \right)^{9/16} \right)^{8/27}} \right)^2 & \text{if } \text{Ra}_L > 10^9 \end{cases} \quad (4-183)$$

where the height of the wall, L , is a correlation input.

The Rayleigh number is defined as:

$$\text{Ra}_L = \frac{g \alpha_p \rho^2 C_p |T - T_{\text{ext}}| L^3}{k \mu} \quad (4-184)$$

if the density depends on temperature only, or as:

$$\text{Ra}_L = \frac{g \rho C_p |\rho_{\text{ext}} - \rho_s| L^3}{k \mu} \quad (4-185)$$

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 $10^4 \leq \text{Ra}_L \leq 10^{13}$. The laminar-turbulent transition at $\text{Ra}_L = 10^9$ is handled by the use of a smoothed Heaviside function with a transition of size 10^8 , which corresponds to 10% of the threshold value.

INCLINED WALL

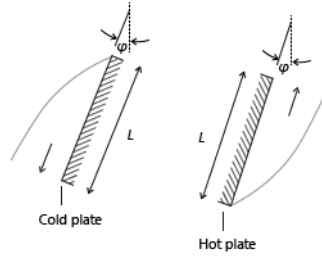


Figure 4-31: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on an inclined wall.

The following correlations correspond to equations 9.26 and 9.27 in Ref. 21 (the same as for a vertical wall):

$$h = \begin{cases} \frac{k}{L} \left(0.68 + \frac{0.67 (\cos \phi \text{Ra}_L)^{1/4}}{\left(1 + \left(\frac{0.492k}{\mu C_p} \right)^{9/16} \right)^{4/9}} \right) & \text{if } \text{Ra}_L \leq 10^9 \\ \frac{k}{L} \left(0.825 + \frac{0.387 \text{Ra}_L^{1/6}}{\left(1 + \left(\frac{0.492k}{\mu C_p} \right)^{9/16} \right)^{8/27}} \right)^2 & \text{if } \text{Ra}_L > 10^9 \end{cases} \quad (4-186)$$

where the length of the wall, L , is a correlation input and ϕ is the tilt angle (the angle between the wall and the vertical direction; $\phi = 0$ for vertical walls). These correlations are valid for $-60^\circ < \phi < 60^\circ$ 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:

$$\text{Ra}_L = \frac{g \alpha_p \rho^2 C_p |T - T_{\text{ext}}| L^3}{k \mu} \quad (4-187)$$

if the density depends on temperature only, or as

$$\text{Ra}_L = \frac{g \rho C_p |\rho_{\text{ext}} - \rho_s| L^3}{k \mu} \quad (4-188)$$

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. 41](#)).



According to [Ref. 21](#), correlations for inclined walls are only satisfactory for the top side of a cold plate or the down face of a hot plate. Hence, these correlations are not recommended for the bottom side of a cold face and for the top side of a hot plate.

The laminar-turbulent transition depends on ϕ (see [Ref. 41](#)). Unfortunately, little data is available about transition. There is some data available in [Ref. 41](#) but this data is only approximate, according to the authors. In addition, data is only provided for water (Pr around 6). For this reason, the flow is defined as turbulent, independently of the ϕ value, when

$$\text{Ra}_L > 10^9$$

All material properties are evaluated at $(T + T_{\text{ext}})/2$, except ρ_s , which is evaluated at the wall temperature, T . The laminar-turbulent transition at $\text{Ra}_L = 10^9$ is handled by the use of a smoothed Heaviside function with a transition of size 10^8 , which corresponds to 10% of the threshold value.

HORIZONTAL PLATE, UPSIDE

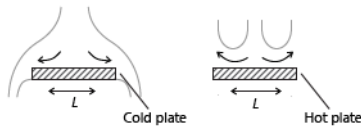


Figure 4-32: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on the top surface of an horizontal plate.

The following correlations correspond to equations 9.30–9.32 in Ref. 21 but can also be found as equations 7.77 and 7.78 in Ref. 41.

If $\rho < \rho_{\text{ext}}$, then

$$h = \begin{cases} \frac{k}{L} 0.54 \text{Ra}_L^{1/4} & \text{if } 10^4 \leq \text{Ra}_L \leq 10^7 \\ \frac{k}{L} 0.15 \text{Ra}_L^{1/3} & \text{if } 10^7 \leq \text{Ra}_L \leq 10^{11} \end{cases} \quad (4-189)$$

while if $\rho \geq \rho_{\text{ext}}$, then

$$h = \frac{k}{L} 0.27 \text{Ra}_L^{1/4} \text{ if } 10^5 \leq \text{Ra}_L \leq 10^{10} \quad (4-190)$$

Ra_L is given by Equation 4-184 or Equation 4-185, and L , the characteristic length (defined as area/perimeter, see Ref. 41) 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 on the temperature only, 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.

The laminar-turbulent transition at $\text{Ra}_L = 10^7$ is handled by the use of a smoothed Heaviside function with a transition of size 10^6 , which corresponds to 10% of the threshold value.

HORIZONTAL PLATE, DOWNSIDE

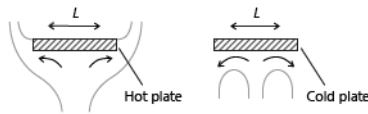


Figure 4-33: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on the bottom surface of an horizontal plate.

Equation 4-189 is used when $\rho \geq \rho_{\text{ext}}$ (or $T \leq T_{\text{ext}}$) and Equation 4-190 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

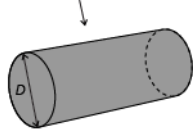


Figure 4-34: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a long horizontal cylinder.

The following correlations correspond to equations 9.34 in [Ref. 21](#). It is validated for $Ra_D \leq 10^{12}$.

$$h = \frac{k}{D} \left(0.6 + \frac{0.387 Ra_D^{1/6}}{\left(1 + \left(\frac{0.559}{Pr} \right)^{9/16} \right)^{8/27}} \right)^2 \quad (4-191)$$

Here D is the cylinder diameter and Ra_D is given by

$$Ra_D = \frac{g \rho C_p \rho^2 |T - T_{\text{ext}}| D^3}{k \mu}$$

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

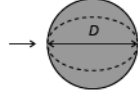


Figure 4-35: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a sphere.

The following correlations correspond to equation 9.35 in Ref. 21. It is validated for $Ra_D \leq 10^{11}$ and $Pr \geq 0.7$.

$$h = \frac{k}{D} \left(2 + \frac{0.589 Ra_D^{1/4}}{\left(1 + \left(\frac{0.469}{Pr} \right)^{9/16} \right)^{4/9}} \right) \quad (4-192)$$

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 .

VERTICAL THIN CYLINDER

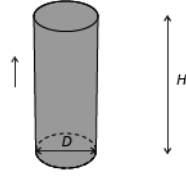


Figure 4-36: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection on a vertical thin cylinder.

The following correlation corresponds to equation 7.83 in Ref. 41. It evaluates the heat transfer coefficient on the lateral surface of the thin cylinder. The horizontal disks (top and bottom) should be treated as horizontal plates.

For the lateral surface, the heat transfer coefficient reads:

$$h = \frac{k}{H} \left(\frac{4}{3} \left(\frac{7 \text{Ra}_H \text{Pr}}{5(20 + 21 \text{Pr})} \right)^{1/4} + \frac{4(272 + 315 \text{Pr})H}{35(64 + 63 \text{Pr})D} \right)$$

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 .

If the thermal boundary layer thickness δ_T is much smaller than the cylinder diameter D , the curvature does not play any significant role, and vertical wall correlations should be used. In practice, the $(\delta_T < D)$ criterion requires

$$\frac{D}{H} > \text{Ra}_H^{-\frac{1}{4}}$$

NARROW CHIMNEY, PARALLEL PLATES

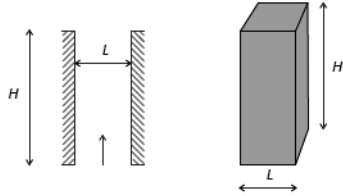


Figure 4-37: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection in a parallel-plate narrow chimney.

If $Ra_L < H/L$, then

$$h = \frac{k}{H} \frac{1}{24} Ra_L \quad (4-193)$$

where the plate distance, L , and the chimney height, H , are correlation inputs (equation 7.96 in Ref. 41). Ra_L is given by Equation 4-184 or Equation 4-185. The material data are evaluated at $(T + T_{\text{ext}})/2$.

NARROW CHIMNEY, CIRCULAR TUBE

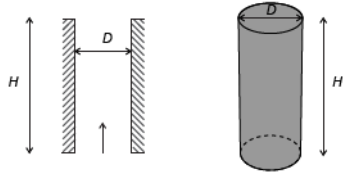


Figure 4-38: Schematic representation of geometry and parameters for the heat transfer coefficient correlation applied to natural convection in a circular narrow chimney.

If $Ra_D < H/D$, then

$$h = \frac{k}{H} \frac{1}{128} Ra_D$$

where the tube diameter, D , and the chimney height, H , are correlation inputs (table 7.2 in Ref. 41 with $D_h = D$). Ra_D is given by Equation 4-184 or Equation 4-185 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

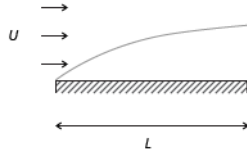


Figure 4-39: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a horizontal plate.

This correlation is a combination of equations 7.33 and 7.41 in Ref. 21:

$$h = \begin{cases} \frac{2k}{L} \frac{0.3387 \text{Pr}^{1/3} \text{Re}_L^{1/2}}{\left(1 + \left(\frac{0.0468}{\text{Pr}}\right)^{2/3}\right)^{1/4}} & \text{if } \text{Re}_L \leq 5 \cdot 10^5 \\ \frac{2k}{L} \text{Pr}^{1/3} (0.037 \text{Re}_L^{4/5} - 871) & \text{if } \text{Re}_L > 5 \cdot 10^5 \end{cases} \quad (4-194)$$

where $\text{Pr} = \mu C_p / k$ and $\text{Re}_L = \rho U L / \mu$. The plate length, L , and the exterior velocity, U , are correlation inputs. The material data are evaluated at $(T + T_{\text{ext}})/2$.

The laminar-turbulent transition at $\text{Re}_L = 5 \cdot 10^5$ is handled by the use of a smoothed Heaviside function with a transition of size $5 \cdot 10^4$, which corresponds to 10% of the threshold value.

PLATE, LOCAL TRANSFER COEFFICIENT

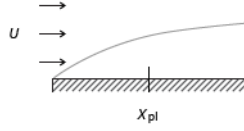


Figure 4-40: Schematic representation of geometry and parameters for the local heat transfer coefficient correlation applied to forced convection on an horizontal plate.

This correlation corresponds to equations 5.79b and 5.131' in Ref. 41:

$$h = \begin{cases} \frac{k}{x} 0.332 \text{Pr}^{1/3} \text{Re}_x^{1/2} & \text{if } \text{Re}_x \leq 5 \cdot 10^5 \\ \frac{k}{x} 0.0296 \text{Pr}^{1/3} \text{Re}_x^{4/5} & \text{if } \text{Re}_x > 5 \cdot 10^5 \end{cases} \quad (4-195)$$

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.

The laminar-turbulent transition at $\text{Re}_x = 5 \cdot 10^5$ is handled by the use of a smoothed Heaviside function with a transition of size $5 \cdot 10^4$, which corresponds to 10% of the threshold value.

CYLINDER IN CROSS-FLOW

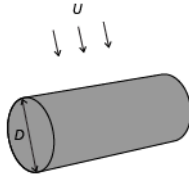


Figure 4-41: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a cylinder in cross-flow.

This correlation is given by equation 7.54 in Ref. 21:

$$h = \frac{k}{D} \left(0.3 + \frac{0.62 \text{Re}_D^{1/2} \text{Pr}^{1/3}}{\left(1 + \left(\frac{0.4}{\text{Pr}} \right)^{2/3} \right)^{1/4}} \left(1 + \left(\frac{\text{Re}_D}{282000} \right)^{5/8} \right)^{4/5} \right) \text{ if } \text{Re}_D \text{Pr} \geq 0.2 \quad (4-196)$$

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

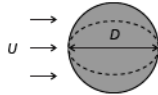


Figure 4-42: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection on a sphere.

This correlation corresponds to equation 7.56 in Ref. 21:

$$h = \frac{k}{D} \left(2 + (0.4 \text{Re}_D^{1/2} + 0.06 \text{Re}_D^{2/3}) \text{Pr}^{0.4} \left(\frac{\mu}{\mu_s} \right)^{1/4} \right) \quad (4-197)$$

$$\text{if } \begin{cases} 3.5 \leq \text{Re}_D \leq 7.6 \cdot 10^4 \\ 0.71 \leq \text{Pr} \leq 380 \\ 1 \leq \frac{\mu}{\mu(T)} \leq 3.2 \end{cases}$$

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 .

Heat Transfer Coefficients — Internal Forced Convection

ISOTHERMAL TUBE

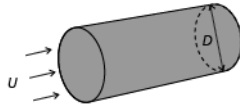


Figure 4-43: Schematic representation of geometry and parameters for the averaged heat transfer coefficient correlation applied to forced convection in an isothermal circular tube.

This correlation corresponds to equations 8.55 and 8.61 in [Ref. 21](#):

$$h = \begin{cases} \frac{k}{D} 3.66 & \text{if } \text{Re}_D \leq 2500 \\ \frac{k}{D} 0.027 \text{Re}_D^{4/5} \text{Pr}^{1/3} \left(\frac{\mu}{\mu_s} \right)^{0.14} & \text{if } \text{Re}_D > 2500 \end{cases} \quad (4-198)$$

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 . The laminar-turbulent transition at $\text{Re}_D = 2500$ is handled by the use of a smoothed Heaviside function with a transition of size 250, which corresponds to 10% of the threshold value.

Using the Heat and Mass Transfer Analogy for the Evaluation of Moisture Transfer Coefficients

In the same way that the heat flux is described by the equation

$$-\mathbf{n} \cdot \mathbf{q} = h(T_{\text{ext}} - T)$$

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

$$-\mathbf{n} \cdot \mathbf{g} = M_v h_m (c_{v, \text{ext}} - c_v) \quad (4-199)$$

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. 21](#), the heat and mass boundary layer analogy is used to express the moisture transfer coefficient h_m as a function of the heat transfer coefficient h :

$$h_m = \frac{D \text{Le}^n}{k} h = \frac{D}{k} \left(\frac{k}{\rho C_p D} \right)^n h \quad (4-200)$$

where Le is the Lewis number, D is the vapor diffusion coefficient in air, k is the thermal conductivity, ρ is the density, and C_p is the specific heat capacity of the external fluid evaluated at the same conditions as those specified in the heat transfer correlation used to compute h .

This relation is valid for laminar and turbulent external flows, and the value $n=1/3$ is suggested as a reasonable assumption for most applications in [Ref. 21](#).

Note that the analogy applies only when the density variations due to temperature and vapor concentration have the same trend. In other words, if the hot air is heavier than cold air due to the moisture, the correlations are not suitable. Else, all the correlations described in [Heat Transfer Coefficients — External Natural Convection](#), [Heat Transfer Coefficients — Internal Natural Convection](#), [Heat Transfer Coefficients — External Forced Convection](#), and [Heat Transfer Coefficients — Internal Forced Convection](#) may be used to evaluate a moisture transfer coefficient h_m to be used in [Equation 4-199](#).

Nucleate Pool Boiling Correlation

Pool boiling is evaporation at a solid surface immersed in a bulk-stationary volume of liquid, where fluid motion is only due to natural convection, by opposition to flow boiling, for which forced convection is considered.

Different regimes of pool boiling may be distinguished, depending on the difference between the surface temperature and the saturation temperature of the boiling fluid:

- When the surface temperature is above the saturation temperature of the fluid by less than 10 K, heat transfer is moderate as it mainly occurs between the surface and the liquid phase of the fluid by convection, due to the formation of isolated bubbles at nucleation sites separating from the surface.
- For a temperature difference above 10 K, more nucleation sites become active. This is called the nucleate boiling regime, at which the heat transfer coefficient can reach much larger values. When temperature difference further increases, it causes interference and coalescence of the bubbles, creation of jets and columns, decreasing fluid motion and convective heat transfer. The critical heat flux value is reached when the effect of these interferences cannot be compensated by an increase of the temperature difference. This happens for a temperature difference of about 30 K for water.

Rohsenow Correlation for Pool Nucleate Boiling

The Rohsenow correlation (Ref. 21) gives an estimation of the heat flux due to nucleate boiling. It reads

$$-\mathbf{n} \cdot \mathbf{q} = -\mu_l L_v \left(\frac{g(\rho_l - \rho_v)}{\sigma} \right)^{1/2} \left[\frac{C_{p,l}(T - T_{\text{sat}})}{\left(\frac{C_{p,l}\mu_l}{k_l} \right)^s C_{\text{sf}} L_v} \right]^3 \quad (4-201)$$

where:

- μ_l , ρ_l , k_l , and $C_{p,l}$ are the viscosity, density, thermal conductivity, and heat capacity of the liquid phase of the fluid, measured at T_{sat}
- ρ_v is the density of the saturated vapor of the fluid
- L_v is the latent heat of vaporization
- g is the gravitational acceleration

- σ is the surface tension of the liquid in contact with its own vapor
- T is the temperature of the solid surface
- T_{sat} is the saturation temperature of the fluid
- s is an empirical constant, equal to 1 for water, and set to 1.7 for other fluids in [Ref. 21](#), [Ref. 41](#), and [Ref. 42](#)
- C_{sf} is an empirical constant accounting for the combination of liquid and surface materials, and for the surface finishing. See [Ref. 21](#), [Ref. 41](#), and [Ref. 42](#) for values of this constant for several combinations.

Note that the correlation is valid for values of gravitational acceleration from 1 to 10 m/s², and for clean surfaces. It is insensitive to the shape and orientation of the surface.

Critical Heat Flux for Nucleate Boiling

The correlation given in [Equation 4-201](#) is valid up to a temperature difference at which the critical heat flux is obtained. The critical heat flux can be expressed as a function of the latent heat, liquid and vapor densities, surface tension, and gravitational acceleration as follows ([Ref. 21](#)):

$$q_{0, \max} = C_{\max} L_v \rho_v^{1/2} [\sigma g (\rho_l - \rho_v)]^{1/4} \quad (4-202)$$

where $C_{\max} = 0.149$ for a large horizontal surface, and $C_{\max} = 0.116$ for a large horizontal cylinder.

Equivalent Thermal Conductivity Correlations

The Nusselt number Nu is the ratio of total heat flux to conductive heat flux. If the fluid flow is not solved, the heat equation can still use an equivalent conductivity to account for the convective heat flux in the conductive part. The conductivity is increased according to Nu to account for the contribution of the convective heat flux.



See [Convectively Enhanced Conductivity](#) node.

Correlations giving Nu from various material properties for two configurations of rectangular enclosures are described below. These correlations are based on the Rayleigh number

$$Ra = \frac{\rho^2 g |\alpha_p| C_p \Delta T L^3}{\mu k}$$

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

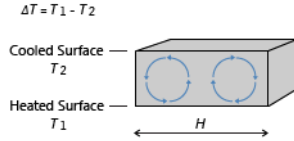


Figure 4-44: Schematic representation of geometry and parameters for the equivalent thermal conductivity correlation applied to an horizontal cavity heated from below.

The following correlation corresponds to equation 9.49 in Ref. 21. It is validated for $3 \cdot 10^5 \leq Ra_H \leq 7 \cdot 10^9$:

$$Nu = 0.069 Ra_H^{1/3} Pr^{0.074} \quad (4-203)$$

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

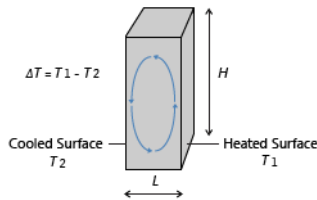


Figure 4-45: Schematic representation of geometry and parameters for the equivalent thermal conductivity correlation applied to a vertical cavity with sidewall heating.

The following correlations correspond to equations 9.50, 9.51, and 9.52 in Ref. 21:

- If $1 \leq H/L \leq 2$, $10^{-3} \leq Pr \leq 10^5$, and $10^3 \leq \frac{Pr Ra_L}{0.2 + Pr}$:

$$\text{Nu} = 0.18 \left(\frac{\text{Pr}}{0.2 + \text{Pr}} \text{Ra}_L \right)^{0.29} \quad (4-204)$$

- If $2 \leq H/L \leq 10$, $\text{Pr} \leq 10^5$, and $10^3 \leq \text{Ra}_L \leq 10^{10}$:

$$\text{Nu} = 0.22 \left(\frac{\text{Pr}}{0.2 + \text{Pr}} \text{Ra}_L \right)^{0.28} \left(\frac{H}{L} \right)^{-1/4} \quad (4-205)$$

- If $10 \leq H/L \leq 40$, $1 \leq \text{Pr} \leq 2 \cdot 10^4$, and $10^4 \leq \text{Ra}_L \leq 10^7$:

$$\text{Nu} = 0.42 \text{Ra}_L^{1/4} \text{Pr}^{0.012} \left(\frac{H}{L} \right)^{-0.3} \quad (4-206)$$

where H is the height of the cavity, L is the distance between the side plates, Ra_L is computed from L and the temperatures T_1 and T_2 of the side walls, and $\text{Pr} = \mu C_p / k$. The material data are evaluated at $(T_1 + T_2)/2$.

Temperature Dependence of Surface Tension

The variation of the surface tension σ (SI unit: N/m) with temperature at fluid interfaces must be taken into account for the computation of phenomena such as the Marangoni effect. These temperature dependencies are available in the form of a coefficient library for a set of liquid-gas interfaces.

For a liquid water-air interface, the following quadratic relation is used:

$$\sigma = -2.3519705 \cdot 10^{-7} T^2 - 1.63350014 \cdot 10^{-5} T + 9.77001279 \cdot 10^{-2}$$

where T (SI unit: K) is the temperature.

In other cases, a linear relation is used instead:

$$\sigma = A(T - T_0) + B$$

where T_0 is a reference temperature taken at 0°C (273.15 K) and the coefficients A (SI unit: N/(m·K)) and B (SI unit: N/m) are given in the following table for some liquid-gas interfaces:

TABLE 4-5: COEFFICIENTS OF SURFACE TENSION.

INTERFACE	A	B	REFERENCE
Acetone - Air	$-1.120 \cdot 10^{-4}$	$2.626 \cdot 10^{-2}$	Ref. 43
Acetic Acid - Air	$-0.994 \cdot 10^{-4}$	$2.958 \cdot 10^{-2}$	Ref. 43
Ethanol - Air	$-0.832 \cdot 10^{-4}$	$2.405 \cdot 10^{-2}$	Ref. 43
Toluene - Air	$-1.189 \cdot 10^{-4}$	$3.09 \cdot 10^{-2}$	Ref. 43
Diethyl Ether - Air	$-0.908 \cdot 10^{-4}$	$1.892 \cdot 10^{-2}$	Ref. 43
Glycerol - Air	$-0.885 \cdot 10^{-4}$	$6.517 \cdot 10^{-2}$	Ref. 43
Heptane - Nitrogen	$-0.980 \cdot 10^{-4}$	$2.21 \cdot 10^{-2}$	Ref. 44
Mercury - Mercury (Vapor)	$-2.049 \cdot 10^{-4}$	$49.06 \cdot 10^{-2}$	Ref. 44
Ethylene Glycol - Ethylene Glycol (Vapor)	$-0.890 \cdot 10^{-4}$	$5.021 \cdot 10^{-2}$	Ref. 44

More data can be found in [Ref. 43](#) and [Ref. 44](#).

Heat Flux and Heat Balance

The concept of heat flux is not as simple as it first might seem. The reason is that heat is not a conserved quantity. Instead, the conserved quantity is the total energy. Hence, there is both a heat flux and an energy flux that are similar but not identical.

This section briefly describes the theory for the variables for [Total Heat Flux and Energy Flux](#), used when computing [Heat and Energy Balance](#). The definitions of these postprocessing variables do not affect the computational results, only variables available for results analysis and visualization.

In this section:

- [Total Heat Flux and Energy Flux](#)
- [Heat and Energy Balance](#)

Total Heat Flux and Energy Flux

TOTAL HEAT FLUX

The total heat flux vector is defined as ([Ref. 5](#)):

$$\mathbf{q}_{\text{tot}} = \rho \mathbf{u}E + \mathbf{q} + \mathbf{q}_r \quad (4-207)$$

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, $\mathbf{q}_{\text{tot}}$, 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. 40) — but it is in general rather difficult to derive. Potential energy is therefore often excluded and the total energy flux is approximated by

$$\mathbf{e}_{\text{tot}} = \rho \mathbf{u} E_0 + \mathbf{q} + \mathbf{q}_r - \sigma \mathbf{u} \quad (4-208)$$

The total energy flux vector, $\mathbf{e}_{\text{tot}}$, 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.

$$\frac{d}{dt} \int_{\Omega} \rho E dv + \int_{\partial\Omega_{\text{ext}}} \mathbf{q}_{\text{tot}} \cdot \mathbf{n} ds = Q_{\text{Int}} - W_{\text{str, Int}} \quad (4-209)$$

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 4-209 is more visually represented by the diagram of [Figure 4-46](#) below.

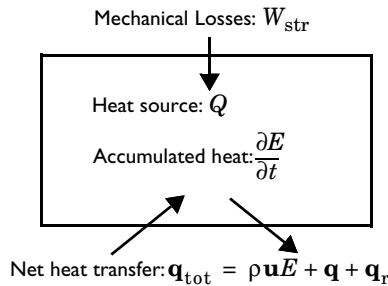


Figure 4-46: A heat balance diagram.

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](#)):

$$\frac{d}{dt} \int_{\Omega} \rho E_0 dv + \int_{\partial\Omega_{\text{ext}}} \mathbf{e}_{\text{tot}} \cdot \mathbf{n} ds = Q_{\text{Int}} + W_{\text{Int}} \quad (4-210)$$

The variables in this formula are defined in [Total Heat Flux and Energy Flux](#). [Equation 4-210](#) is more visually represented by the diagram of [Figure 4-47](#) below.

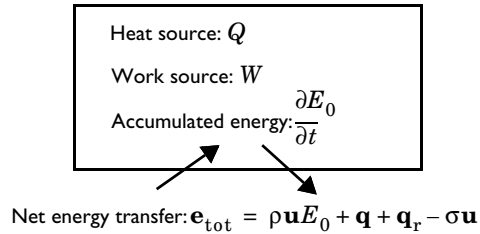


Figure 4-47: An energy balance diagram.

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, all equations are solved on the spatial frame, with the exception of the **Solid**, **Biological Tissue**, **Building Material**, **Shape Memory Alloy** and **Battery Layers** features for which they are solved in the material frame. A set of variables is always defined in the spatial frame, and a second set is also defined in the material frame for the features cited above. 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*:

$$\mathbf{F} = \begin{bmatrix} \frac{\partial x}{\partial X} & \frac{\partial x}{\partial Y} & \frac{\partial x}{\partial Z} \\ \frac{\partial y}{\partial X} & \frac{\partial y}{\partial Y} & \frac{\partial y}{\partial Z} \\ \frac{\partial z}{\partial X} & \frac{\partial z}{\partial Y} & \frac{\partial z}{\partial Z} \end{bmatrix} \quad (4-211)$$

It relates elementary distances $d\mathbf{x}$ and $d\mathbf{X}$ in the domain, expressed in material and spatial frames, according to:

$$d\mathbf{x} = \mathbf{F}d\mathbf{X} \quad (4-212)$$

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.

Most of 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:

$$\int_{\Omega_0} \rho_{(X,Y,Z)} d\tau_0 = \int_{\Omega} \rho_{(x,y,z)} \det(\mathbf{F}) d\tau = \int_{\Omega} \rho_{(x,y,z)} d\tau$$

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 4-212](#).

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

$$\nabla_{(X,Y,Z)} T = \mathbf{F}^T \nabla_{(x,y,z)} T$$

and that the unit normal vector $\mathbf{n}$ verify that $\mathbf{n}_{(X,Y,Z)}$ and $\mathbf{F}^T \mathbf{n}_{(x,y,z)}$ are collinear, 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.

When the material frame differs from the spatial frame, the domain features representing solid domains give the option to specify the thermal conductivity either on the spatial or the material frame. See [Heat Conduction, Solid](#) for details about this option.

THERMAL CONDUCTIVITY OF A LAYER

The same transformations are applied to thermal conductivity but with different transformation matrices. The deformation gradient tensor depends on the layer type:

- When the layer is resistive, the deformation gradient tensor $\mathbf{F}_{\text{xdim}}$ is equal to the deformation gradient tensor $\mathbf{F}$ defined in Equation 4-211.
- When the layer is conductive, the deformation gradient tensor $\mathbf{F}_{\mathbf{t}}$ is defined using tangential derivatives as follows:

$$\mathbf{F}_{\mathbf{t}} = \begin{bmatrix} xT_X & xT_Y & xT_Z \\ yT_X & yT_Y & yT_Z \\ zT_X & zT_Y & zT_Z \end{bmatrix}$$

where xT_X corresponds to the tangential derivative x with respect to X , and so on.

- When the layer is an extra dimension, the deformation gradient tensor $\mathbf{F}_{\text{xdim}}$ 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. Hasgall, F. Di Gennaro, C. Baumgartner, E. Neufeld, M.C. Gosselin, D. Payne, A. Klingensböck, N. Kuster, *IT’IS Database for thermal and electromagnetic parameters of biological tissues*, Version 3.0, 2015. <https://itis.swiss/virtual-population/tissue-properties/overview/>

12. C. Rossmann and D. Haemmerich, “Review of Temperature Dependence of Thermal Properties, Dielectric Properties, and Perfusion of Biological Tissues at Hyperthermic and Ablation Temperatures,” *Crit. Rev. Biomed. Eng.*, vol. 42, pp. 467–492, 2014.
13. D.A. Nield and A. Bejan, *Convection in Porous Media*, in Convection Heat Transfer, Fourth Edition, John Wiley & Sons, Inc., Hoboken, NJ, USA, 2013.
14. B. Sundén and J. Yuan, “Evaluation of Models of the Effective Thermal Conductivity of Porous Materials Relevant to Fuel Cell Electrodes,” *Int. J. Comp. Meth. and Exp. Meas.*, vol. 1, no. 4, 2013.
15. N. Wakao, S. Kaguei, and T. Funazkri, “Effect of fluid dispersion coefficients on particle-to-fluid heat transfer coefficients in packed beds,” *Chem. Eng. Sci.*, vol. 34, pp. 325–336, 1979.
16. EN 15026, *Hygrothermal performance of building components and building elements - Assessment of moisture transfer by numerical simulation*, CEN, 2007.
17. J. Bear and Y. Bachmat, *Introduction to Modeling of Transport Phenomena in Porous Media*, Kluwer Academic Publisher, 1990.
18. H. Lee, *Thermal Design, Heat Sinks, Thermoelectrics, Heat Pipes, Compact Heat Exchangers, and Solar Cells*, John Wiley & Sons, 2006.
19. G. Fraisse, J. Ramousse, D. Sgorlon, and C. Goupil, “Comparison of different modeling approaches for thermoelectric elements,” *Energy Convers. Manag.*, vol. 65, pp. 351–356, 2013.
20. R. Siegel and J. Howell, *Thermal Radiation Heat Transfer*, 4th ed., Taylor & Francis, New York, 2002.
21. F.P. Incropera, D.P. DeWitt, T.L. Bergman, and A.S. Lavine, *Fundamentals of Heat and Mass Transfer*, 6th ed., John Wiley & Sons, 2006.
22. <https://gml.noaa.gov/grad/solcalc/>
23. M.F. Modest, *Radiative Heat Transfer*, 2nd ed., Academic Press, San Diego, California, 2003.
24. W.A. Fiveland, “The Selection of Discrete Ordinate Quadrature Sets for Anisotropic Scattering,” *Fundamentals of Radiation Transfer*, HTD, vol. 160, ASME, 1991.
25. A. Bejan and others, *Heat Transfer Handbook*, John Wiley & Sons, 2003.

26. F. Charron, *Partage de la chaleur entre deux corps frottants*, Publication Scientifique et Technique du Ministère de l'Air, no. 182, 1943. (In French)
27. D.C. Wilcox, *Turbulence Modeling for CFD*, 2nd ed., DCW Industries, 1998.
28. J. Larsson, *Numerical Simulation of Turbulent Flows for Turbine Blade Heat Transfer*, Doctoral Thesis for the Degree of Doctor of Philosophy, Chalmers University of Technology, Sweden, 1998.
29. W.M. Kays, "Turbulent Prandtl Number — Where Are We?," *ASME J. Heat Transfer*, vol. 116, pp. 284–295, 1994.
30. B. Weigand, J.R. Ferguson, and M.E. Crawford, "An Extended Kays and Crawford Turbulent Prandtl Number Model," *Int. J. Heat and Mass Transfer*, vol. 40, no. 17, pp. 4191–4196, 1997.
31. D. Lacasse, È. Turgeon, and D. Pelletier, "On the Judicious Use of the k - ϵ Model, Wall Functions and Adaptivity," *Int. J. Thermal Sciences*, vol. 43, pp. 925–938, 2004.
32. D.A. Nield, "Effects of local thermal non-equilibrium in steady convective processes in a saturated porous medium: forced convection in a channel," *J. Porous Media*, vol. 1, pp. 181–186, 1998.
33. W.J. Minkowycz and others, "On departure from local thermal equilibrium in porous media due to a rapidly changing heat source: the Sparrow number," *Int. J. Heat Mass Transf.*, vol. 42, pp. 3373–3385, 1999.
34. A. Amiri and K. Vafai, "Transient analysis of incompressible flow through a packed bed," *Int. J. Heat Mass Transf.*, vol. 41, pp. 4259–4279, 1998.
35. R.G. Carbonell and S. Whitaker, "Heat and Mass Transfer in Porous Media," *Fundamentals of Transport Phenomena in Porous Media*, J. Bear and M.Y. Corapcioglu, eds., Springer, 1984, pp. 121–198.
36. J.L. Monteith and M.H. Unsworth, *Principles of Environmental Physics*, Edward Arnold, London, 290 pp., 1990.
37. P.T. Tsilingiris, "Thermophysical and Transport Properties of Humid Air at Temperature Range Between 0 and 100°C," *Energy Convers. Manag.*, vol. 49, no. 5, pp. 1098–1110, 2008.
38. J. Zhang, A. Gupta, and J. Bakera, "Effect of Relative Humidity on the Prediction of Natural Convection Heat Transfer Coefficients," *Heat Transf. Eng.*, vol. 28, no. 4, pp. 335–342, 2007.

39. W. Wagner and H-J Kretzschmar, *International Steam Tables*, 2nd ed., Springer, 2008.
40. M. Kaviany, *Principles of Convective Heat Transfer*, 2nd ed., Springer, 2001.
41. A. Bejan, *Heat Transfer*, John Wiley & Sons, 1993.
42. I. Piore, “Experimental Evaluation of Constants for the Rohsenow Pool Boiling Correlation,” *Int. J. Heat Mass Transf.*, vol. 42, no. 11, pp. 2003–2013, 1999.
43. Tables of Physical & Chemical Constants (16th edition 1995). 2.2.5 Surface tension. Kaye & Laby Online. Version 1.0 (2005), www.kayelaby.npl.co.uk
44. J.J. Jasper, “The Surface Tension of Pure Liquid Compounds”, *J. Phys. Chem. Ref. Data*, vol. 1, pp. 841–1010, 1972.
45. G.K. Batchelor, *An Introduction to Fluid Dynamics*, Cambridge University Press, 2000.
46. C.P. Thurgood, A. Pollard, and H.A. Becker, “The T_N quadrature set for the discrete ordinates method,” *J. Heat Transfer*, vol. 117, pp. 1068–1070, 1995.
47. M.A. Badri, P. Jolivet, B. Rousseau, S. Le Corre, H. Digonnet and Y. Favennec, “Vectorial finite elements for solving the radiative transfer equation,” *J. Quant. Spectrosc. Radiat. Transf.*, vol. 212, pp. 59–74, 2018.
48. R.R. Bate and others, *Fundamentals of astrodynamics*, Courier Dover Publications, 2020.
49. M.Th. van Genuchten, “A Closed-form Equation for Predicting the Hydraulic Conductivity of Unsaturated Soils,” *Soil Sci. Soc. Am. J.*, vol. 44, 1980.
50. R.H. Brooks and A.T. Corey, “Properties of Porous Media Affecting Fluid Flow,” *J. Irrig. Drainage Div., ASCE Proc.*, vol. 72 (IR2), 1966.

The Heat Transfer Module Interfaces

The Heat Transfer Module includes several physics interfaces used to compute the temperature field, the radiative intensity field, or the relative humidity field:

TABLE 5-1: THE HEAT TRANSFER MODULE INTERFACES.

INTERFACE	TAG	VERSIONS
Heat Transfer	ht	Heat Transfer in Solids
		Heat Transfer in Fluids
		Heat Transfer in Solids and Fluids
		Bioheat Transfer
		Heat Transfer in Porous Media
		Local Thermal Nonequilibrium
		Heat Transfer in Packed Beds
		Heat Transfer in Moist Porous Media
		Heat Transfer in Building Materials
		Heat Transfer in Moist Air
Heat Transfer in Shells	htlsh	Heat Transfer in Shells
		Heat Transfer in Films
		Heat Transfer in Fractures

TABLE 5-1: THE HEAT TRANSFER MODULE INTERFACES.

INTERFACE	TAG	VERSIONS
Lumped Thermal System	lts	Lumped Thermal System
Surface-to-Surface Radiation	rad	Surface-to-Surface Radiation
Orbital Thermal Loads	otl	Orbital Thermal Loads
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 Equilibrium Moisture Transport in Porous Media Nonequilibrium Moisture Transport in Porous Media Moisture Transport in Building Materials Moisture Transport in Free and Porous Media
Moisture Transport in Solids	mts	Moisture Transport in Solids

See also [The Heat Transfer in Pipes Interface](#) ([http](#)) in the *Pipe Flow Module User's Guide*.

This chapter begins with a quick presentation of the interfaces of the Heat Transfer Module:

- [The Heat Transfer Interfaces](#)
- [The Conjugate Heat Transfer Interfaces](#)
- [The Radiation Interfaces](#)
- [The Electromagnetic Heating Interfaces](#)
- [The Thin Structures Interfaces](#)
- [The Heat and Moisture Transport Interfaces](#)
- [The Heat and Moisture Flow Interfaces](#)
- [The Moisture Transport Interfaces](#)
- [The Moisture Flow Interfaces](#)

- The Porous Media Interfaces
- The Metal Processing Interfaces
- The Curing Interfaces

More details about the physics interfaces of the Heat Transfer Module are presented in the following sections:

- About the Heat Transfer Interfaces
- The Heat Transfer in Solids Interface
- The Heat Transfer in Fluids Interface
- The Heat Transfer in Solids and Fluids Interface
- The Heat Transfer in Porous Media Interface
- The Local Thermal Nonequilibrium Interface
- The Heat Transfer in Packed Beds Interface
- The Heat Transfer in Moist Porous Media Interface
- The Heat Transfer in Building Materials Interface
- The Heat Transfer in Moist Air Interface
- The Bioheat Transfer Interface
- The Heat Transfer in Shells Interface
- The Heat Transfer in Films Interface
- The Heat Transfer in Fractures Interface
- The Lumped Thermal System Interface
- The Surface-to-Surface Radiation Interface
- The Orbital Thermal Loads 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 Equilibrium Moisture Transport in Porous Media Interface
- The Nonequilibrium Moisture Transport in Porous Media Interface
- The Moisture Transport in Building Materials Interface
- The Moisture Transport in Free and Porous Media Interface
- The Moisture Transport in Solids Interface

See also the [Multiphysics Interfaces](#).

The Heat Transfer Interfaces

Under the Heat Transfer group, physics and multiphysics interfaces are either directly under the group node, or organized in subgroups:

- [The Heat Transfer in Solids Interface](#)
- [The Heat Transfer in Fluids Interface](#)
- [The Heat Transfer in Solids and Fluids Interface](#)
- [The Conjugate Heat Transfer Interfaces](#)
- [The Radiation Interfaces](#)
- [The Electromagnetic Heating Interfaces](#)
- [The Lumped Thermal System Interface](#)
- [The Thin Structures Interfaces](#)
- [The Heat and Moisture Transport Interfaces](#)
- [The Porous Media Interfaces](#)
- [The Metal Processing Interfaces](#)
- [The Bioheat Transfer Interface](#)
- [The Curing Interfaces](#)
- [The Heat Transfer in Pipes Interface](#)
- [The Thermoelectric Effect Interface](#)

The following sections quickly present the interfaces directly under the group node.

THE HEAT TRANSFER IN SOLIDS INTERFACE

By default, the Heat Transfer in Solids interface () describes heat transfer by conduction. It can also account for heat flux due to translation in solids (for example, the rotation of a disk or the linear translation of a shaft) as well as for solid deformation, including volume or surface changes. In the case of irreversible thermally induced transformation, it accounts for the enthalpy and material properties changes.



See [The Heat Transfer in Solids Interface](#) for details.

THE HEAT TRANSFER IN FLUIDS INTERFACE

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



See [The Heat Transfer in Fluids Interface](#) for details.

THE HEAT TRANSFER IN SOLIDS AND FLUIDS INTERFACE

The Heat Transfer in Solids and Fluids interface () contains solids and fluids domains by default. It is aimed to simplify the setup of models where capabilities of Heat Transfer in Solids interface () and Heat Transfer in Fluids interface () are used, in particular in conjugate heat transfer applications.



See [The Heat Transfer in Solids and Fluids Interface](#) for details.

THE LUMPED THERMAL SYSTEM INTERFACE

The Lumped Thermal System interface () extends the heat transfer modeling possibility to the discrete thermal systems. The external terminal feature connects a lumped thermal system to a finite element model in any dimension. This is particularly helpful to reduce dramatically the models complexity, for example to describe thermal interaction between parts in large assemblies. Several classical devices like conductive or radiative thermal resistor are predefined. In addition advanced devices, heat pipe and thermoelectric module, and user's defined subsystems are available.



See [The Lumped Thermal System Interface](#) for details.

THE BIOHEAT TRANSFER INTERFACE

The Bioheat Transfer interface () is a dedicated interface for heat transfer in living tissue. In addition to data such as thermal conductivity, heat capacity, and density, tabulated data is available for blood perfusion rates and metabolic heat sources. Tissue damage integral models based on a temperature threshold or an energy absorption model can also be included.



See [The Bioheat Transfer Interface](#) for details.

THE HEAT TRANSFER IN PIPES INTERFACE

The Heat Transfer in Pipes interface () accounts for conduction and convection heat transfer. It is used when the velocity and pressure fields of the fluid flow are known a priori and the pipes can be represented by a 1D line with a constant cross-sectional shape. This reduces the complexity to a 1D model and can provide the temperature field or thermal losses along the pipe.



See [The Heat Transfer in Pipes Interface](#) for details.

THE THERMOELECTRIC EFFECT INTERFACE

The Thermoelectric Effect interface () combines the Electric Currents and the Heat Transfer in Solids interfaces with capabilities for modeling thermoelectric effects (Peltier-Seebeck-Thomson effects) as well as Joule heating (resistive heating). This multiphysics coupling accounts for Peltier heat source or sink and resistive losses in the Heat Transfer interfaces as well as for the current induced by the Seebeck effect and for the temperature dependency of material properties in the Electric Currents interface. This physics interface is automatically paired with the AC/DC module capabilities for advanced modeling of electric effects.



See [About The Thermoelectric Effect Interface](#) for details.

The Conjugate Heat Transfer Interfaces

The Conjugate Heat Transfer interfaces () combine all features from the Heat Transfer and Single-Phase Flow interfaces to describe heat transfer in solids and fluids and nonisothermal flow in fluids. The heat transfer process is tightly coupled with the fluid flow problem via a predefined multiphysics coupling. These interfaces are available for laminar and turbulent nonisothermal flow and flow in porous media (Brinkman equation):

- [The Laminar Flow Interface](#)
- [The Turbulent Flow Interfaces](#)

THE LAMINAR FLOW INTERFACE

For simulations of heat transfer between a solid and a fluid in the laminar flow regime, the Laminar Flow interface () combines a Heat Transfer in Solids interface with a Laminar Flow interface. The temperature is resolved all the way to the solid wall which ensures accurate simulations in this flow regime.



See [The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces](#) for details.

THE TURBULENT FLOW INTERFACES

For highly accurate simulations of heat transfer between a solid and a fluid in the turbulent flow regime, low-Reynolds turbulence models resolve the temperature field in the fluid all the way to the solid wall. This model is available in the Turbulent Flow, Low-Re k - ϵ interface () as well as in the Turbulent Flow, SST interface (). The SST (shear stress transport) interface is suitable for many external flow cases and internal flows with sudden expansion. The standard k - ϵ turbulence model in the Turbulent Flow, k - ϵ interface () is computationally inexpensive compared to the other transport two-equation turbulence models but usually less accurate. The Algebraic y Plus and L-VEL interfaces are adapted for internal flows.

With the use of the CFD Module, additional turbulence models are available. The Realizable k - ϵ model is similar to the standard k - ϵ model but has built-in realizability constraints, resulting in improved performance for certain flows, such as turbulent jets.

The $k\text{-}\omega$ model is an alternative to the standard $k\text{-}\epsilon$ model and often gives more accurate results, especially in recirculation regions and close to solid walls. However, the $k\text{-}\omega$ model is also less robust than the standard $k\text{-}\epsilon$ model. The Spalart-Allmaras interface is a dedicated physics interface for conjugate heat transfer in aerodynamics, for example, in the simulation of wing profiles. The v2-f model includes near-wall turbulence anisotropy and is giving highly accurate results.



See [The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces](#) for details.

The Radiation Interfaces

The Radiation interfaces () for heat transfer essentially belong to two different groups of radiation modeling: surface-to-surface radiation and radiation in participating or absorbing media:

- The Heat Transfer with Surface-to-Surface Interface
- The Heat Transfer with Orbital Thermal Loads 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

In cases where the temperature is an input of the model, more interfaces are available to compute only the radiative effects:

- The Surface-to-Surface Radiation Interface
- The Orbital Thermal Loads Interface
- The Radiation in Participating Media Interface
- The Radiation in Absorbing-Scattering Media Interface
- The Radiative Beam in Absorbing Media Interface

THE HEAT TRANSFER WITH SURFACE-TO-SURFACE INTERFACE

The Heat Transfer with Surface-to-Surface Radiation multiphysics interface () combines the Heat Transfer in Solids interface with the Surface-to-Surface Radiation interface. It includes all functionality to model heat transfer in fluids or solids, including conduction and convection with surface-to-surface radiation. The surface-to-surface radiation model also accounts for the dependency of surface properties on the spectral bands. For example, to model the greenhouse effect, it is necessary to solve separately for ambient radiation (large wavelengths) and the sun's radiation (small wavelengths). In addition, specular reflection can be considered instead of diffuse reflection. This is of particular interest for glossy surfaces made of polished aluminum or silver for example. Finally semitransparent layers can be modeled

for configuration where a fraction of the radiation is going through a layer while the rest is reflected.



See [The Heat Transfer with Surface-to-Surface Radiation Multiphysics Interface](#) for details.

THE HEAT TRANSFER WITH ORBITAL THERMAL LOADS INTERFACE

The Heat Transfer with Orbital Thermal Loads multiphysics interface () combines the Heat Transfer in Solids interface with the Orbital Thermal Loads interface. It is used to model heat transfer including conduction in solid parts and radiative loads on spacecraft, in particular radiation from the Sun and the Earth for satellites orbiting around Earth.



See [The Heat Transfer with Orbital Thermal Loads Multiphysics Interface](#) for details.

THE HEAT TRANSFER WITH RADIATION IN PARTICIPATING MEDIA INTERFACE

The Heat Transfer with Radiation in Participating Media multiphysics interface () combines the Heat Transfer in Solids interface with the Radiation in Participating Media interface. It includes all functionality to model conduction and convection in solids and fluids with radiation where absorption, emission, and scattering of radiation is accounted for by the radiation model.



See [The Heat Transfer with Radiation in Participating Media Multiphysics Interface](#) for details.

THE HEAT TRANSFER WITH RADIATION IN ABSORBING-SCATTERING MEDIA INTERFACE

For media that can be considered as nonemitting, the Heat Transfer with Radiation in Absorbing-Scattering Media multiphysics interface () combines the Heat Transfer

in Solids interface with the Radiation in Absorbing-Scattering Media interface. For example, emission may be neglected when considering light diffusion.



See [The Heat Transfer with Radiation in Absorbing-Scattering Media Multiphysics Interface](#) for details.

THE HEAT TRANSFER WITH RADIATIVE BEAM IN ABSORBING MEDIA INTERFACE

For the modeling of collimated radiative beam propagating with no refraction, no reflection, and no scattering in a semitransparent medium that is supposed not to emit in the wavelength of the incident light, the Heat Transfer with Radiative Beam in Absorbing Media multiphysics interface () combines the Heat Transfer in Solids interface with the Radiative Beam in Absorbing Media interface.



See [The Heat Transfer with Radiative Beam in Absorbing Media Multiphysics Interface](#) for details.

THE SURFACE-TO-SURFACE RADIATION INTERFACE

The Surface-to-Surface Radiation interface () describes systems where only radiation is computed, typically to estimate radiation between surfaces in space applications where the surface temperature is known.



See [The Surface-to-Surface Radiation Interface](#) for details.

THE ORBITAL THERMAL LOADS INTERFACE

The Orbital Thermal Loads interface () is used to compute radiation received by systems in orbit around a celestial body.



See [The Orbital Thermal Loads Interface](#) for details.

THE RADIATION IN PARTICIPATING MEDIA INTERFACE

The Radiation in Participating Media interface () computes the radiation, with the possibility to include absorption, emission, and scattering effects, in a medium where the temperature is known.



See [The Radiation in Participating Media Interface](#) for details.

THE RADIATION IN ABSORBING-SCATTERING MEDIA INTERFACE

The Radiation in Absorbing-Scattering Media interface () computes the radiation, with the possibility to include absorption, emission, and scattering effects, in a medium where the temperature is known.



See [The Radiation in Absorbing-Scattering Media Interface](#) for details.

THE RADIATIVE BEAM IN ABSORBING MEDIA INTERFACE

The Radiative Beam in Absorbing Media interface () computes the radiation, with the possibility to include absorption, emission, and scattering effects, in a medium where the temperature is known.



See [The Radiative Beam in Absorbing Media Interface](#) for details.

The Electromagnetic Heating Interfaces

The Electromagnetic Heating interfaces () combine a Heat Transfer in Solids interface with an interface of the AC/DC module. It models the heating in a solid produced by electromagnetism, and handles the temperature dependency of the electromagnetic characteristics of the material.

In this subgroup:

- [The Joule Heating Interface](#)
- [The Induction Heating Interface](#)
- [The Microwave Heating Interface](#)
- [The Laser Heating Interface](#)

THE JOULE HEATING INTERFACE

The Joule Heating interface () combines the Electric Currents and Heat Transfer in Solids interfaces to model Joule heating (resistive heating). This multiphysics coupling accounts for electromagnetic losses in the Heat Transfer interfaces as well as for the temperature dependency of material properties in the Electric Currents interface.



See [The Joule Heating Interface](#) for details.

THE INDUCTION HEATING INTERFACE

The Induction Heating interface () combines the Magnetic Fields and Heat Transfer in Solids interfaces to model induction heating. This multiphysics coupling accounts for electromagnetic losses in the Heat Transfer interfaces as well as for the temperature dependency of material properties in the Magnetic Fields interface.



See [The Induction Heating Interface](#) for details.

THE MICROWAVE HEATING INTERFACE

The Microwave Heating interface () combines the Electromagnetic Waves, Frequency Domain and Heat Transfer in Solids interfaces to model microwave heating. This multiphysics coupling accounts for electromagnetic losses in the Heat Transfer interfaces as well as for the temperature dependency of material properties in the Electromagnetic Waves, Frequency Domain interface,



See [The Microwave Heating Interface](#) for details.

THE LASER HEATING INTERFACE

The Laser Heating interface () combines the Electromagnetic Waves, Beam Envelopes and Heat Transfer in Solids interfaces to model microwave heating. This multiphysics coupling accounts for electromagnetic losses in the Heat Transfer interfaces as well as for the temperature dependency of material properties in the Electromagnetic Waves, Beam Envelopes interface,



See [The Laser Heating Interface](#) for details.

The Thin Structures Interfaces

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

- [The Heat Transfer in Shells Interface](#)
- [The Heat Transfer in Films Interface](#)
- [The Heat Transfer in Fractures Interface](#)

THE HEAT TRANSFER IN SHELLS INTERFACE

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



See [The Heat Transfer in Shells Interface](#) for details.

THE HEAT TRANSFER IN FILMS INTERFACE

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

the sides of the film are exposed to different temperature or when heat is dissipated in the film.



See [The Heat Transfer in Films Interface](#) for details.

THE HEAT TRANSFER IN FRACTURES INTERFACE

The Heat Transfer in Fractures interface () describes heat transfer in a thin porous media. The simplest model assumes that the temperature changes through the fracture thickness can be neglected, while the general model computes the temperature variation across the fracture.



See [The Heat Transfer in Fractures Interface](#) for details.

The Heat and Moisture Transport Interfaces

The Heat and Moisture Transport interfaces () are multiphysics interfaces coupling heat transfer with moisture transport. Several interfaces are available to study the transport of heat and moisture in different media:

- [The Moist Air Interface](#)
- [The Moist Porous Media interface](#)
- [The Building Materials Interface](#)
- [The Heat and Moisture Flow Interfaces](#)

THE MOIST AIR INTERFACE

The Moist Air interface () combines the Heat Transfer in Moist Air interface with the Moisture Transport in Air interface. It is used to simulate the coupling between heat transfer and vapor transport in air and the evaporation and condensation on walls.



See [The Moist Air Version of the Heat and Moisture Transport Multiphysics Interface](#) for details.

THE MOIST POROUS MEDIA INTERFACE

The Moist Porous Media interface () combines the Heat Transfer in Moist Porous Media and Equilibrium Moisture Transport in Porous Media interfaces. It is used to simulate the coupling between heat transfer and moisture transport in liquid and gas phases in the pores of the medium.



See [The Moist Porous Media Version of the Heat and Moisture Transport Multiphysics Interface](#) for details.

THE BUILDING MATERIALS INTERFACE

The Building Materials interface () combines the Heat Transfer in Building interface with the Moisture Transport in Building Materials interface. It can be used to model different moisture variations phenomena in building components such as drying of initial construction moisture, condensation due to migration of moisture

from outside to inside, or moisture accumulation by interstitial condensation due to diffusion.



See [The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface](#) for details.

The Heat and Moisture Flow Interfaces

The Heat and Moisture Flow interfaces () combine all features from the Heat Transfer in Moist Air, Moisture Transport in Air, and Single-Phase Flow interfaces. Three multiphysics couplings — Heat and Moisture Flow, Nonisothermal Flow, and Moisture Flow — are also automatically added.

The Heat and Moisture Flow interfaces () are used to compute the temperature and relative humidity distribution in air along with the velocity and pressure fields. This multiphysics coupling is available for laminar flow, turbulent flow, and flow in porous media (Brinkman equation):

- [The Laminar Flow interface](#)
- [The Turbulent Flow Interfaces](#)
- [The Porous Media Flow Interface](#)

THE LAMINAR FLOW INTERFACE

The Laminar Flow interface () uses the laminar version of the Single-Phase Flow interface to compute the velocity and the pressure inside the fluid domain. It is suitable for a laminar flow regime.



See [The Heat and Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces](#) for details.

THE TURBULENT FLOW INTERFACES

The Turbulent Flow interfaces () uses a version of the Single-Phase Flow interface with a turbulence model to compute the velocity and the pressure inside the fluid domain. Several turbulence models are available, with different levels of accuracy and computational costs. These interfaces are suitable for a turbulent flow regime.



See [The Heat and Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces](#) for details.

THE POROUS MEDIA FLOW INTERFACE

The Porous Media Flow interfaces () uses a Brinkman Equations interface to compute the velocity and the pressure inside the fluid domain. It is suitable for a flow in a porous medium.



See [The Heat and Moisture Flow, Brinkman Equations Interface](#) for details.

The Moisture Transport Interfaces

The Moisture Transport interfaces () are used to compute the relative humidity in a medium. It comes in several versions to account for different media:

- [The Moisture in Air Interface](#)
- [The Equilibrium Moisture Transport in Porous Media Interface](#)
- [The Nonequilibrium Moisture Transport in Porous Media Interface](#)
- [The Moisture Transport in Building Materials Interface](#)
- [The Moisture Transport in Free and Porous Media Interface](#)
- [The Moisture Transport in Solids Interface](#)

THE MOISTURE IN AIR INTERFACE

The Moisture Transport in Air interface () is used to compute the relative humidity distribution in air. It simulates moisture transport by vapor convection and diffusion in moist air and the evaporation or condensation on walls.



See [The Moisture Transport in Air Interface](#) for details.

THE EQUILIBRIUM MOISTURE TRANSPORT IN POROUS MEDIA INTERFACE

The Equilibrium Moisture Transport in Porous Media interface () is used to compute the relative humidity field in porous media. It simulates moisture transport by vapor convection and diffusion in the gas phase, liquid water transport by convection and capillarity in the pores of the media. A Hygroscopic Porous Medium feature is active by default on all domains.



See [The Equilibrium Moisture Transport in Porous Media Interface](#) for details.

THE NONEQUILIBRIUM MOISTURE TRANSPORT IN POROUS MEDIA INTERFACE

The Nonequilibrium Moisture Transport in Porous Media interface () is used to compute the relative humidity field and liquid water saturation in porous media. It

simulates moisture transport by vapor convection and diffusion in the gas phase, liquid water transport by convection and capillarity in the pores of the media. A Hygroscopic Porous Medium feature is active by default on all domains. It differs from simpler macroscale models for moisture transport in porous media where the amount of water in vapor and liquid phases follows the equilibrium condition.



See [The Nonequilibrium Moisture Transport in Porous Media Interface](#) for details.

THE MOISTURE TRANSPORT IN BUILDING MATERIALS INTERFACE

The Moisture Transport in Building Materials interface () is used to compute the relative humidity field in building materials. It simulates moisture transport by taking into account moisture storage, liquid transport by capillary suction forces, and convective transport of vapor.



See [The Moisture Transport in Building Materials Interface](#) for details.

THE MOISTURE TRANSPORT IN FREE AND POROUS MEDIA INTERFACE

The Moisture Transport in Free and Porous Media interface () is used to compute the relative humidity field in all domains, and additionally the liquid water saturation in porous media. It simulates moisture transport by vapor convection and diffusion in the gas phase, liquid water transport by convection and capillarity, in the pores of the media. A Moist Air feature is active by default on all domains, and a Hygroscopic Porous Medium is added by default with no selection. This interface handles all types of interface between free and porous media, and between two porous media.



See [The Moisture Transport in Free and Porous Media Interface](#) for details.

THE MOISTURE TRANSPORT IN SOLIDS INTERFACE

The Moisture Transport in Solids interface () is used to compute the relative humidity and moist air pressure in porous media. It simulates moisture transport by vapor convection and diffusion in the gas phase and liquid water transport by convection and capillarity in the pores of the medium. A Porous Medium feature is active by default on all domains. This interface has its own specific features, and

formulation of equations, making it different from the other Moisture Transport interfaces.



See [The Moisture Transport in Solids Interface](#) for details.

The Moisture Flow Interfaces

The Moisture Flow interfaces () combine all features from the Moisture Transport in Air and Single-Phase Flow interfaces. The Moisture Flow multiphysics coupling is automatically added. It couples the moisture transfer and fluid flow interfaces. The fluid properties may depend on vapor concentration. Models can also include moisture transport in building materials. The physics interface supports low Mach numbers (typically less than 0.3). Similarly to Conjugate Heat Transfer coupling, the moisture transport interface can be coupled with one of the turbulent flow interface.

If a module providing the Brinkman Equation interface is available, the Porous Media Flow coupling is also available. It couples moisture transport in porous media and fluid flow by Brinkman equations.

In this subgroup:

- [The Laminar Flow Interface](#)
- [The Turbulent Flow Interfaces](#)
- [The Porous Media Flow Interface](#)

THE LAMINAR FLOW INTERFACE

The Laminar Flow interface () couples the moisture transport in air with the fluid flow when the flow regime is laminar. This interface can be used either for a free flow or a flow in porous media.



See [The Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces](#) for details.

THE TURBULENT FLOW INTERFACES

The Turbulent Flow interfaces () couple the moisture transport in air with the fluid flow when the flow regime is turbulent. It comes in several versions with different turbulence models.



See [The Moisture Flow, Laminar Flow and Turbulent Flow Multiphysics Interfaces](#) for details.

THE POROUS MEDIA FLOW INTERFACE

The Porous Media Flow interface () couples the moisture transport in air and porous media with the fluid flow. A Brinkman Equation interface is used to resolve the velocity and pressure fields in the porous media.



See [The Moisture Flow, Brinkman Equations Multiphysics Interface](#) for details.

The Porous Media Interfaces

The Porous Media interfaces () simulate heat transfer in porous media combining a liquid and a solid phase. The interfaces correspond to the different assumptions of heat transfer behavior between the phases:

- [The Heat Transfer in Porous Media Interface](#)
- [The Local Thermal Nonequilibrium Interface](#)
- [The Heat Transfer in Packed Beds Interface](#)

THE HEAT TRANSFER IN POROUS MEDIA INTERFACE

The Heat Transfer in Porous Media interface () combines conduction in a porous matrix and in the fluid contained in the pore structure with the convection of heat generated by the flow of the fluid. This physics interface uses the provided power law or a user-defined expression for the effective heat transfer properties, and a predefined expression for dispersion in porous media. Dispersion is caused by the tortuous path of the liquid in the porous media. (This would be absent if the mean convective term was accounted for.) This physics interface may be used for a wide range of porous materials, from porous structures in the pulp and paper industry to the simulation of heat transfer in soil and rocks.



See [The Heat Transfer in Porous Media Interface](#) for details.

THE LOCAL THERMAL NONEQUILIBRIUM INTERFACE

The Local Thermal Nonequilibrium (LTNE) interface () implements a macroscale model designed to simulate heat transfer in porous media where the temperatures in the porous matrix and the fluid are not in equilibrium. It differs from simpler macroscale models for heat transfer in porous media where the temperature difference between the solid and fluid is neglected. The absence of thermal equilibrium can result from fast transient changes, but it can also be observed in stationary cases. Typical applications are rapid heating of a porous media using a hot fluid or internal heat generation in one of the phases (due to inductive or microwave heating, exothermic

reactions, and so on). This is observed in nuclear devices, electronics systems, or fuel cells for example.



See [The Local Thermal Nonequilibrium Interface](#) for details.

THE HEAT TRANSFER IN PACKED BEDS INTERFACE

The Heat Transfer in Packed Beds interface () provides a multiscale model for heat transfer in a porous medium where the local thermal equilibrium is not assumed between the pellets of a packed bed and the fluid phase, and where the temperature variation inside the pellets is taken into account. A temperature field is defined for each phase, pellets and fluid of the porous medium, and the heat transfer between them is accounted for. The microscale pellets temperature field depends on the radial coordinate of each pellet.



See [The Heat Transfer in Packed Beds Interface](#) for details.

The Metal Processing Interfaces

The Metal Processing interfaces () study the structure of a metal undergoing a heat process. The phase fractions of the metal are computed from an initial state. In specific interfaces, a thermal loading can be applied on the model, and the resulting deformation of the heat process is also available.

In this subgroup:

- [The Metal Phase Transformation Interface](#)
- [The Austenite Decomposition Interface](#)
- [The Heat Transfer with Phase Transformations Interface](#)
- [The Steel Quenching Interface](#)
- [The Carburization Interface](#)

THE METAL PHASE TRANSFORMATION INTERFACE

The Metal Phase Transformation interface () resolves each phase fraction of a metal undergoing a heat process.



See [The Metal Phase Transformation Interface](#) for details.

THE AUSTENITE DECOMPOSITION INTERFACE

The Austenite Decomposition interface () resolves each phase fraction of a metal undergoing a heat process. It is typically intended to compute the resulting structure of a metal during a cooling process from an austenitic state.



See [The Austenite Decomposition Interface](#) for details.

THE HEAT TRANSFER WITH PHASE TRANSFORMATIONS INTERFACE

The Heat Transfer with Phase Transformations interface () is a multiphysics coupling between a Heat Transfer in Solids interface and a Metal Phase

Transformation interface. It computes both the temperature and the phase fractions of a metal during a thermal loading.



See [Heat Transfer with Phase Transformations](#) for details.

THE STEEL QUENCHING INTERFACE

The Steel Quenching interface () is a multiphysics couplings between a Heat Transfer in Solids interface, a Solid Mechanics interface, and an Austenite Decomposition interface. It is typically used to compute the resulting structure and deformation of the metal during quenching.



See [Steel Quenching](#) for details.

THE CARBURIZATION INTERFACE

The Carburization interface () is intended for modeling carburization processes during heat treatment.



See [The Carburization Interface](#) for details.

The Curing Interfaces

The Curing interfaces () computes the kinetics of the curing reaction in polymer during heat treatment.

THE CURING INTERFACE

The Curing Interface () resolves the conversion of the curing reaction. Under heat treatment, polymers undergo a curing reaction, which kinetics can be parameterized in this interface.



See [The Curing Reaction Interface](#) for details.

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
- Orbital Thermal Loads
- 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 Interface

The versions of the Heat Transfer interface (ht) are:

- [The Heat Transfer in Solids Interface](#)
- [The Heat Transfer in Fluids Interface](#)
- [The Heat Transfer in Solids and Fluids Interface](#)
- [The Bioheat Transfer Interface](#)
- [The Heat Transfer in Porous Media Interface](#)
- [The Local Thermal Nonequilibrium Interface](#)
- [The Heat Transfer in Moist Porous Media Interface](#)
- [The Heat Transfer in Building Materials Interface](#)
- [The Heat Transfer in Moist Air Interface](#)

After selecting a version, default nodes are added under the main node, which then defines which version of the Heat Transfer interface is added. Depending on the version of the physics interface selected, the default nodes vary. For example:

- If **Heat Transfer in Solids** () is selected, a **Heat Transfer in Solids (ht)** interface is added with a default **Solid** model.

- If **Heat Transfer in Fluids** () is selected, a **Heat Transfer in Fluids (ht)** interface is added with a default **Fluid** model.
- If **Heat Transfer in Solids in Fluids** () is selected, a **Heat Transfer in Solids and Fluids (ht)** interface is added with two default models: **Solid** (enabled by default) and **Fluid** (empty selection by default).

Benefits of the Different Heat Transfer Interfaces

The benefit of the different versions of the Heat Transfer interface, with ht as the common default name (see [Heat Transfer Variables](#)), is that it is easy to add the default settings when selecting the physics interface. At any time, add a **Fluid** or **Solid** node from the **Physics** toolbar — the functionality is always available.



Depending on the available COMSOL products, a checkbox 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 Interface

The versions of the Heat Transfer in Shells interface (ht1sh) 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 (ht1sh)** interface is added with a default **Solid** model.
- If **Heat Transfer in Films** () is selected, a **Heat Transfer in Films (ht1sh)** interface is added with a default **Fluid** model.
- If **Heat Transfer in Fractures** () is selected, a **Heat Transfer in Fractures (ht1sh)** interface is added with a default **Porous Medium** model.

Benefits of the Different Heat Transfer in Shells Interfaces

The benefit of the different versions of the Heat Transfer in Shells interface, with `htlsh` as the common default name (see [Heat Transfer Variables](#)), is that it is easy to add the default settings when selecting the physics interface. At any time, add a **Solid** node from the **Physics** toolbar — the functionality is always available.

TABLE 5-2: THE HEAT TRANSFER IN SHELLS (HTLSH) INTERFACE OPTIONS.

ICON	NAME	DEFAULT PHYSICAL MODEL
	Heat Transfer in Shells	No checkbox is selected.
	Heat Transfer in Films	No checkbox is selected.
	Heat Transfer in Fractures	The Heat transfer in porous media checkbox 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.

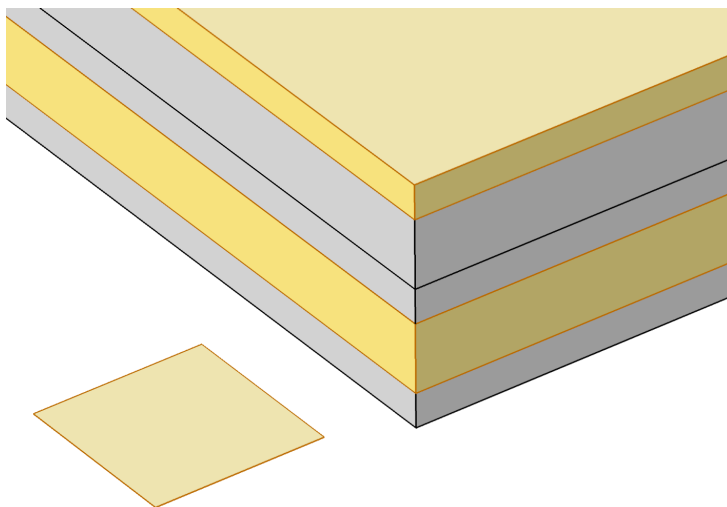


Figure 5-1: Boundary selection with layers 2 and 5 selected.

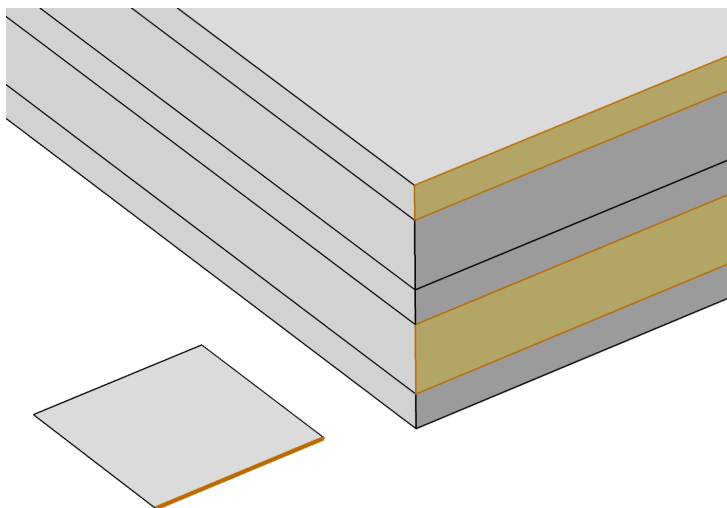


Figure 5-2: Edge selection with layers 2 and 5 selected.

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 checkboxes 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.

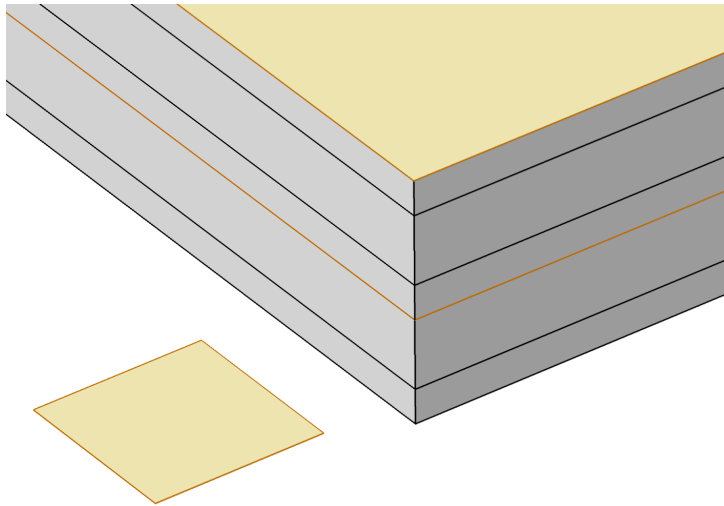


Figure 5-3: Boundary selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

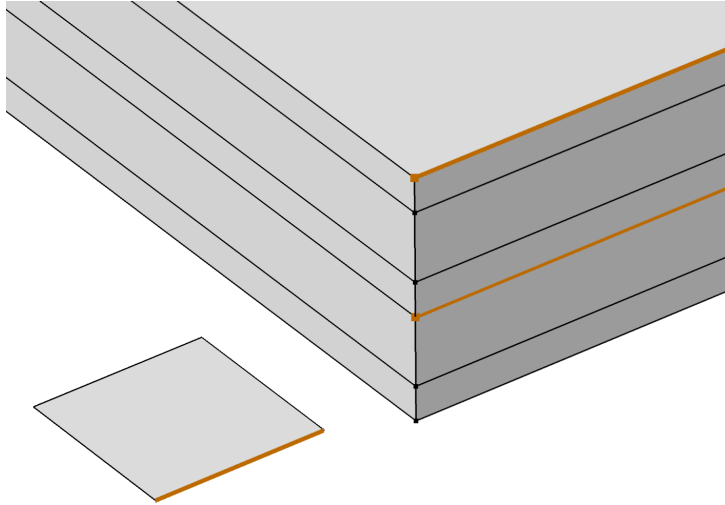


Figure 5-4: Edge selection with interfaces 'layer 2 - layer 3' and 'layer 5 up' selected.

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.

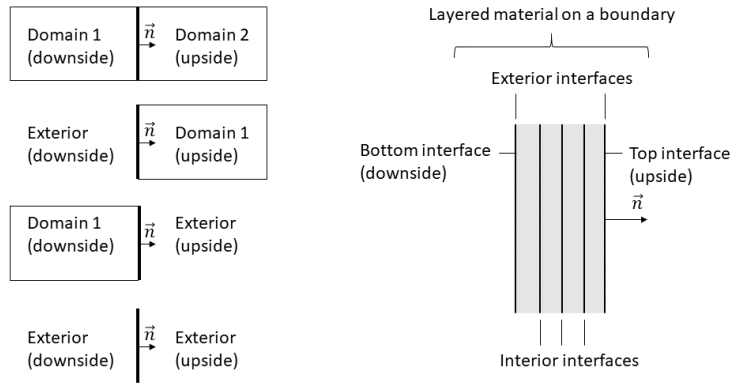


Figure 5-5: Definition of upside and downside settings for a boundary for different configurations, and corresponding interfaces for a layered material.



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_1lmat1_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 Interface

The versions of the Moisture Transport interface (`mt`) are:

- [The Moisture Transport in Air Interface](#)

- The Equilibrium Moisture Transport in Porous Media Interface
- The Nonequilibrium Moisture Transport in Porous Media Interface
- The Moisture Transport in Building Materials Interface
- The Moisture Transport in Free and Porous Media 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 **Equilibrium Moisture Transport in Porous Media** () is selected, an **Equilibrium Moisture Transport in Porous Media (mt)** interface is added with a default **Hygroscopic Porous Medium** model. The latter is set to **Equilibrium formulation**.
- If **Nonequilibrium Moisture Transport in Porous Media** () is selected, a **Nonequilibrium Moisture Transport in Porous Media (mt)** interface is added with a default **Hygroscopic Porous Medium** model. The latter is set to **Nonequilibrium formulation**.
- 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 Free and Porous Media** () is selected, a **Moisture Transport in Free and Porous Media (mt)** interface is added with default **Moist Air** and **Hygroscopic Porous Medium** models. The latter is set to **Nonequilibrium formulation**.

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**, a **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, SST, 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)	—
	Heat and Moisture	—
	Thermoelectric Effect	—

More turbulent flow interfaces are available under the **Conjugate Heat Transfer** branch with the CFD Module:

- Turbulent Flow, Realizable k - ϵ
- Turbulent Flow, k - ω
- Turbulent Flow, Spalart-Allmaras
- Turbulent Flow, v_2 - 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 h_{Tref} .



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.

The **Streamline diffusion** checkbox 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** checkbox is also selected by default and is only available when the **Streamline diffusion** checkbox is selected.

INCONSISTENT STABILIZATION

The **Isotropic diffusion** checkbox 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. 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
- Cell Periodicity
- Convectively Enhanced Conductivity
- Cross Section
- Fluid
- Fluid (Porous Medium)
- Geothermal Heating
- Heat Source
- Heat Source (Pellets)
- Immobile Fluids (Porous Medium)
- Initial Values
- Irreversible Transformation
- Isothermal Domain
- Liquid Water (Moist Porous Medium)
- Moist Air (Heat Transfer Interface)
- Moist Air (Moist Porous Medium)
- Moist Porous Medium
- Optically Thick Participating Medium
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Pellets (Porous Medium)
- Pellet-Fluid Interface (Porous Medium)
- Phase Change Material
- Porous Matrix (Porous Medium, Moist Porous Medium)
- Porous Medium
- Pressure Work
- Shape Memory Alloy
- Solid
- Thermal Damage
- Thermal Dispersion
- Thermoelastic Damping
- Thickness
- Translational Motion
- Viscous Dissipation

BOUNDARY NODES FOR THE HEAT TRANSFER INTERFACE

The Heat Transfer interface has the following boundary nodes:

- Boundary Heat Source
- Boundary Pair
- Continuity
- Deposited Beam Power
- External Temperature (Thin Layer, Thin Film, Fracture)
- Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
- Heat Flux
- Heat Source (Thin Layer, Thin Film, Fracture)
- Inflow
- Line Heat Source on Axis
- Local Thermal Nonequilibrium Boundary
- Lumped System Connector
- Open Boundary
- Outflow
- Periodic Condition (Heat Transfer Interface)
- Phase Change Interface
- Surface-to-Ambient Radiation (Heat Transfer Interface)
- Symmetry (Heat Transfer Interface)
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)

When an **Isothermal Domain** feature is present, 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** checkbox 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** checkbox 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 checkboxes corresponding to layers where the interface should not be applied in the **Selection** table.

Nonlayered Shell

If the **Restrict to layered boundaries** checkbox 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, <code>ht1sh.ds</code>, used in the weak equations, is the product of the thickness set in the Shell Properties section, <code>ht1sh.lth</code>, and the scale factor <code>ht1sh.lsc</code>, 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, <code>ht1sh.ds</code>, is defined in all dimensions, and is unrelated to the Out-of-plane thickness of the layered material, <code>ht1sh.d</code>, 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 <i>COMSOL Multiphysics Reference Manual</i> 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 4-73](#)). 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 <code>ht.Tref</code> variable.

CONSISTENT STABILIZATION

The **Streamline diffusion** checkbox 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** checkbox is also selected by default and is only available when the **Streamline diffusion** checkbox is selected.

INCONSISTENT STABILIZATION

The **Isotropic diffusion** checkbox 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. 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, the Moisture Transport in Building Materials, and the Moisture Transport in Free and Porous Media interfaces. 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, the compressibility effects are neglected.
- For larger vapor concentration conditions, the variation of moist air density are modeled, and the **Concentrated species** formulation should be used. This is the default option in the Equilibrium Moisture Transport in Porous Media and the Nonequilibrium Moisture Transport in Porous Media interfaces. In this case, the compressibility effects are taken into account.

Note that this setting impacts only moist air domains, and has no effect on building materials and hygroscopic porous media domains.

Finally, select the **Account for gravity in liquid velocity** checkbox to enable the [Gravity](#) feature.

CONSISTENT STABILIZATION

The **Streamline diffusion** checkbox 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** checkbox is also selected by default and is only available when the **Streamline diffusion** checkbox is selected.

INCONSISTENT STABILIZATION

The **Isotropic diffusion** checkbox is not selected by default. To add isotropic diffusion, select the **Isotropic diffusion** checkbox. The field for the tuning parameter δ_{id} then becomes available. The default value is 0.25; increase or decrease the value of δ_{id} to increase or decrease the amount of isotropic diffusion.



In the *COMSOL Multiphysics Reference Manual*:

- [Stabilization Techniques](#)
- [Selection Information](#)

ADVANCED SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options**. These settings only apply to the domains in which a **Moist Air** feature is active. Select a **Convective term** — **Nonconservative form** (the default) or **Conservative form**. The latter should be used for compressible flow.

Note that **Diluted Species** and **Concentrated Species** formulations are equivalent when using **Conservative form**, they can both be used for large vapor concentrations.

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. The shape functions used for the relative humidity are by default **Quadratic Lagrange** in building materials, and **Linear** in moist air and porous media. Note that when a porous medium

is used with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation**, an additional dependent variable is solved for, the liquid saturation, which uses the same type of shape functions as the relative humidity.



Theory for Moisture Transport

Feature Nodes for the Moisture Transport Interface

This section details the nodes available with the Moisture Transport interface.

DOMAIN NODES FOR THE MOISTURE TRANSPORT INTERFACE

The Moisture Transport interface has the following domain nodes:

- Building Material
- Hygroscopic Porous Medium
- Initial Values
- Liquid Water (Hygroscopic Porous Medium)
- Moist Air (Hygroscopic Porous Medium)
- Moist Air (Moisture Transport Interface)
- Moisture Source
- Porous Matrix (Hygroscopic Porous Medium)
- Turbulent Mixing

BOUNDARY NODES FOR THE MOISTURE TRANSPORT INTERFACE

The Moisture Transport interface has the following boundary nodes:

- Continuity
- Inflow
- Insulation
- Liquid Water Content
- Liquid Water Flux
- Moist Surface
- Moisture Content
- Moisture Flux
- Nonequilibrium Boundary
- Open Boundary
- Outflow
- Periodic Condition
- Porous Interface
- Symmetry
- Thin Moisture Barrier
- Vapor Content
- Vapor Flux
- Wet Surface

GLOBAL NODES FOR THE MOISTURE TRANSPORT INTERFACE

The Moisture Transport interface has one global nodes: [Gravity](#).

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 in the **Porous Media** physics area under the **Heat Transfer** branch () when adding a physics interface, is used to model heat transfer by conduction and convection in porous media. A **Porous Medium** model is active by default on all domains, with **Porous medium type** set to **Local thermal equilibrium**. All functionality for including other domain types, such as a solid domain, is also available, and surface-to-ambient radiation may be considered.

The temperature equation defined in porous media domains corresponds to the convection-diffusion equation with thermodynamic properties averaging models to account for both solid matrix and fluid properties. This equation is valid when the temperatures into the porous matrix and the fluid are in equilibrium. If not, use [The Local Thermal Nonequilibrium Interface](#) instead.

The physics interface is an extension of the generic Heat Transfer interface. When this physics interface is added, the following default nodes are added in the **Model Builder**: **Porous Medium**, **Thermal Insulation** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Heat Transfer in Porous Media** to select physics features from the context menu.

By default, the shape functions used for the temperature in porous media are **Linear**.

	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.
	Theory for Heat Transfer in Porous Media
	<i>Free Convection in a Porous Medium</i> : Application Library path Subsurface_Flow_Module/Heat_Transfer/convection_porous_medium

The Local Thermal Nonequilibrium Interface

The **Local Thermal Nonequilibrium** interface (), found in the **Porous Media** physics area under the **Heat Transfer** branch () when adding a physics interface, is used to model heat transfer by conduction and convection, in porous media for which the solid and fluid phase temperatures are not in equilibrium. A **Porous Medium** model is active by default on all domains, with **Porous medium type** set to **Local thermal nonequilibrium**. All functionality for including other domain types, such as a solid domain, is also available, and surface-to-ambient radiation may be considered.

The temperature equations in the solid and fluid phases of the porous medium are solved and coupled through a transfer term proportional to the temperature difference between both phases. If this difference can be neglected, use [The Heat Transfer in Porous Media Interface](#) instead.

The physics interface is an extension of the generic Heat Transfer interface. When this physics interface is added, the following default nodes are added in the **Model Builder**: **Porous Medium**, **Thermal Insulation** (the default boundary condition) overridden by **Local Thermal Nonequilibrium Boundary** (showing all the boundaries adjacent to domains where two temperatures are solved for solid and fluid phases), and **Initial Values**.

Specific subnodes are also present by default under the **Fluid** and **Porous Matrix** subnodes of the **Porous Medium** node:

- **Initial Values**, which allows to set a specific initial temperature for each phase
- **Thermal Insulation**, which sets a no flux condition for each phase
- **Continuity** (under **Fluid** subnode only), which ensures the continuity of the temperature between the fluid phase and the other domains.

Other subnodes implementing boundary conditions specific to the fluid and solid phases can be added to model flow conditions, heat sources, fluxes, and phase change. To do so, right-click a **Fluid** or **Porous Matrix** node to select physics features from the context menu.

Heat Sources

In the heat source features available under the **Fluid** and **Porous Matrix** subnodes, the user input corresponds to the heat production per total unit volume. It is multiplied by the volume fraction of each phase and added into the corresponding heat equation.

Heat Fluxes

In the heat flux features available under the **Fluid** and **Porous Matrix** subnodes, the user input corresponds to the heat flux per total unit surface. It is multiplied by the volume fraction of each phase and added into the corresponding heat equation. The surface fraction is approximated by the volume fraction.

By default, the shape functions used for the temperature in porous media are **Linear**.

	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.
	Theory for Heat Transfer in Porous Media

The Heat Transfer in Packed Beds Interface

The **Heat Transfer in Packed Beds** interface (), found in the **Porous Media** physics area under the **Heat Transfer** branch () when adding a physics interface, is used to model heat transfer by conduction and convection, in porous media where the local thermal equilibrium is not assumed between the solid pellets of a packed bed and the fluid phase, and where the radial variation of the temperature due to conduction inside the pellets is accounted for. A **Porous Medium** model is active by default on all domains, with **Porous medium type** set to **Packed bed**. All functionality for including other domain types, such as a solid domain, is also available, and surface-to-ambient radiation may be considered, assuming that radiation effects are handled on the exterior surface of the pellets only.

The microscale heat equation in the pellets and the macroscale heat equation in the fluid are solved and coupled either through a transfer term proportional to the temperature difference between both phases, or a continuity condition on the fluid and pellets temperatures.

The physics interface is an extension of the generic Heat Transfer interface. When this physics interface is added, the following default nodes are added in the **Model Builder**: **Porous Medium**, **Thermal Insulation** (the default boundary condition) overridden by **Local Thermal Nonequilibrium Boundary** (showing all the boundaries adjacent to domains where two temperatures are solved for solid and fluid phases), and **Initial Values**.

Specific subnodes are also present by default under the **Fluid** and **Pellets** subnodes of the **Porous Medium** node:

- **Initial Values**, which allows to set a specific initial temperature for each phase
- **Thermal Insulation** (under **Fluid** subnode only), which sets a no flux condition for the fluid phase
- **Continuity** (under **Fluid** subnode only), which ensures the continuity of the temperature between the fluid phase and the other domains.

Other subnodes implementing boundary conditions specific to the fluid and pellets may be added, to model flow conditions, heat sources, fluxes, and phase change. This

can be done by right-clicking **Fluid** or **Pellets** to select physics features from the context menu.

By default, the shape functions used for the temperature in porous media are **Linear**.

	See Settings for the Heat Transfer Interface and Feature Nodes for the Heat Transfer Interface for details.
	Theory for Heat Transfer in Porous Media

The Heat Transfer in Moist Porous Media Interface

The **Heat Transfer in Moist Porous Media (ht)** interface () 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 <i>COMSOL Multiphysics Reference Manual</i> 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
	<i>Hepatic Tumor Ablation</i> : 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 4-73](#)) 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. 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. 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.
	• 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:

- 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 SHELLS INTERFACE

The Heat Transfer in Shells Interface has the following boundary interface nodes:

- Deposited Beam Power, Interface (Heat Transfer in Shells Interface)
- Heat Flux, Interface (Heat Transfer in Shells Interface)
- Heat Source, Interface (Heat Transfer in Shells Interface)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in Shells Interface)
- Thermal Insulation, 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)
- Lumped System Connector (Heat Transfer in Shells Interface)
- Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

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. 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. 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:

- 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 FILMS INTERFACE

The [Heat Transfer in Films Interface](#) has the following boundary interface nodes:

- Deposited Beam Power, Interface (Heat Transfer in Shells Interface)
- Heat Flux, Interface (Heat Transfer in Shells Interface)
- Heat Source, Interface (Heat Transfer in Shells Interface)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in Shells Interface)
- Thermal Insulation, 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)
- Lumped System Connector (Heat Transfer in Shells Interface)
- Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

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. 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. 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)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in Shells Interface)
- Thermal Insulation, Interface (Heat Transfer in Shells Interface)

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)
- Lumped System Connector (Heat Transfer in Shells Interface)
- Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

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 <i>COMSOL Multiphysics Reference Manual</i> 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.

- [Events Timeline](#)
- [Convective Thermal Resistor](#)
- [Heat Pipe](#)
- [Heat Rate Source](#)
- [Radiative Thermal Resistor](#)
- [Thermal Capacitor](#)
- [Thermoelectric Module](#)

The following nodes are available from the **Physics** toolbar, **Global** menu or from by selecting from the **Subsystems** subnode after right-clicking the **Lumped Thermal System** interface node.

- [Subsystem Definition](#)
- [Subsystem Instance](#)

The following nodes are available from the **Physics** toolbar, **Global** menu or from by selecting from the **Nodes** subnode after right-clicking the **Lumped Thermal System** interface node.

- [External Terminal](#)
- [Heat Rate](#)
- [Radiative Heat Rate](#)
- [Temperature](#)
- [Thermal Mass](#)

The Surface-to-Surface Radiation Interface

The **Surface-to-Surface Radiation (rad)** interface () , found under the **Heat Transfer > Radiation** branch () , is used to model heat transfer by radiation. It treats thermal radiation as an energy transfer between boundaries and external heat sources where the medium does not participate in the radiation (transparent medium).



This interface does not compute the temperature field, it requires it as model input. If the medium participates in the radiation (semitransparent medium), then use [The Radiation in Participating Media Interface](#) instead.

The radiosity method is used on boundaries where **Surface-to-Surface Radiation** is enabled.

From the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Surface-to-Surface Radiation** to select physics features from the context menu. For the **Surface-to-Surface Radiation** interface, select a **Stationary** or **Time Dependent** study as a preset study type. **Surface-to-Surface Radiation** is always stationary (that is, the radiation time scale is assumed to be shorter than any other time scale), but the physics interface is compatible with all standard study types.



Absolute (thermodynamical) temperature units must be used. See [Using Expressions and Variables](#) 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](#)
- [Predefined Functions 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 be defined for each spectral band. In particular it is possible to define the solar absorptivity for short wavelengths and the surface emissivity for large wavelengths. For integrations calculation, the upper bound of the spectral band for ambient radiation is predefined and set to 1 [mm]. This value can be modified by selecting the **Specify the upper bound for integrations** checkbox.
- Select **Multiple spectral bands** and set the value of the **Right endpoint** for each spectral band in the table, to define a diffuse spectral radiation model. The value of the **Right endpoint** must be entered without unit. Modify the **Wavelength unit** to set the unit of all the endpoints. **Right endpoint** values should be set in an ascending order. The values of the **Left endpoint** for the next spectral band are updated automatically. It is possible to provide a definition of the surface properties for each spectral band.



The first **Left endpoint** and the last **Right endpoint** are predefined and equal to 0 and $+\infty$, respectively.

Parameters can be used to set the value of the **Right endpoint**.



The wavelengths λ set in **Solar and ambient** and **Multiple spectral bands** are the wavelengths in vacuum.

Modify the **Transparent media refractive index** to define the refractive index. Vacuum has a refractive index of 1, which is usually a good approximation for air refractive index.

Select the **Use radiation groups** checkbox to be able to define [Radiation Group \(Surface-to-Surface Radiation Interface\)](#). Radiation groups can be used to speed up the radiation calculations by gathering boundaries that can see one another.

Select the **Surface-to-surface radiation method: Direct area integration, Hemicube** (the default), or **Ray shooting**:

- For [Direct Area Integration](#) select a **Radiation integration order** — **4** is the default.
- For [Hemicube](#) select a **Radiation resolution** — **256** is the default.
- For [Ray Shooting](#) select a **Radiation resolution** — **8** is the default.

TABLE 5-4: SURFACE-TO-SURFACE RADIATION METHODS.

	DIRECT AREA INTEGRATION	HEMICUBE	RAY SHOOTING
Supported dimensions	3D, 2Daxi, 2D, 1D axi, 1D	3D, 2Daxi, 2D, 1D axi, 1D	3D, 2D, 2Daxi
Shadowing effects		√	√
Diffuse reflection	√	√	√
Specular reflection			√
Refraction			√
Directional properties			√
Wavelength-dependent properties	√ (Solar and ambient / Multiple spectral bands)	√ (Solar and ambient / Multiple spectral bands)	√ (Solar and ambient / Multiple spectral bands)
External radiation source	√	√	√

TABLE 5-4: SURFACE-TO-SURFACE RADIATION METHODS.

	DIRECT AREA INTEGRATION	HEMICUBE	RAY SHOOTING
Symmetry	√	√	√
View factor update control	√	√	√

Hemicube

Hemicube is the default method for the heat transfer interfaces. The more sophisticated and general hemicube method uses a *z*-buffered projection on the sides of a hemicube (with generalizations to 2D and 1D) to account for shadowing effects. Think of it as rendering digital images of the geometry in five different directions (in 3D; in 2D only three directions are needed), and counting the pixels in each mesh element to evaluate its view factor.

Its accuracy can be influenced by setting the **Radiation resolution** of the virtual snapshots. The number of *z*-buffer pixels on each side of the 3D hemicube equals the specified resolution squared. Thus the time required to evaluate the irradiation increases quadratically with resolution. In 2D, the number of *z*-buffer pixels is proportional to the resolution property, and thus the time is, as well.

Select the **Use adaptive resolution coarsening** checkbox to control if resolution coarsening is used. Groups of four adjacent pixels are then merged as one when the difference of irradiation between these four pixels is below the **Tolerance** (default value is 1e-3). It is a recursive process that can result in only one pixel on each side of the hemicube. By reducing the resolution, this option reduces the memory required to solve the model.

For an axisymmetric geometry, G_m and F_{amb} must be evaluated in a corresponding 3D geometry obtained by revolving the 2D boundaries about the axis. COMSOL Multiphysics creates this virtual 3D geometry by revolving the 2D boundary mesh into a 3D mesh. The resolution can be controlled in the azimuthal direction by setting the number of **Azimuthal sectors**, which is the same as the number of elements to a full revolution. Try to balance this number against the mesh resolution in the *rz*-plane.

Direct Area Integration

COMSOL Multiphysics evaluates the mutual irradiation between surface directly, without considering which face elements are obstructed by others. This means that shadowing effects (that is, surface elements being obstructed in nonconvex cases) are not taken into account. Elements facing away from each other are, however, excluded from the integrals.

Direct area integration is fast and accurate for simple geometries with no shadowing, or where the shadowing can be handled by manually assigning boundaries to different radiation groups.



If shadowing is ignored, global energy is not conserved. Control the accuracy by specifying a **Radiation integration order**. Sharp angles and small gaps between surfaces may require a higher integration order for accuracy but also more time to evaluate the irradiation.

Ray Shooting

The ray shooting algorithm accounts for directional properties in the view factor evaluation. This includes specular reflection and transmission of radiation and also the angular dependence of surface properties. Select this method to enable the [Opaque Surface \(Surface-to-Surface Radiation Interface\)](#), [Specular Mirror \(Surface-to-Surface Radiation Interface\)](#) and [Semitransparent Surface \(Surface-to-Surface Radiation Interface\)](#) boundary conditions.

To compute the radiation incident upon surfaces, the ray shooting algorithm uses a backward ray tracing approach, emitting rays from each element outward, to determine the total irradiation from surrounding elements. For each element, $n+1$ rays are launched in 2D and n^2+1 rays in 3D, where n is the value selected for **Radiation resolution**. To determine the direction of these rays, the hemisphere (angular space) is divided into n tiles in 2D and n^2 in 3D. The trajectories are computed as the rays are absorbed, reflected, or transmitted on the model surfaces until their intensity becomes too small or if the rays go far away from the geometry. The threshold where the ray trajectory is no longer computed is controlled by the **Tolerance**. On their trajectory, two adjacent rays can hit boundaries belonging to two different features, or detects differences in the radiosity. In such cases, the number of tiles can be locally refined up to a number of times defined by the **Maximum number of adaptations**. This aims to detect irradiation from boundaries that are smaller than the tile size.

Set the **Maximal number of reflections** (default value is 1000) to define the maximal number of times that a specific ray is allowed to be reflected by a surface. This avoids infinite loops in case of perfectly reflecting surfaces. Using large values improves the view factor accuracy while small ones can be used to limit the view factor evaluation time.

When using directional dependent surface properties, set **Angular dependent properties** to define how the selected directional functions are evaluated.

- Keep the default value, **Full resolution**, to evaluate the directional function for each direction. This option improves accuracy and is the most computationally intensive.
- Select **Interpolated function** to sample the directional function on a grid with $(k+1)$ points for Polar functions and $(k+1)^2$ grid points for Polar and Azimuthal functions. k is the **Number of interpolation points** (default value is 100), a strictly positive integer. The evaluation is done using linear interpolation between the sampled values. This option can speed up the view factor evaluation when directional dependent surface properties are used.

In 2D and 3D, when the **Use forward ray shooting for external radiation source** checkbox is selected, the contribution of **External Radiation Source** features to the irradiation is calculated using a forward ray tracing approach. Set the **Number of forward rays** (default value is 100000) to define the number of rays launched by each **External Radiation Source** feature. This option is expected to improve accuracy when rays from **External Radiation Source** are reflected several times.



The forward ray shooting approach for external radiation source is not supported with symmetries and cavities. In that case, the backward ray shooting for external radiation source is always used.

To improve the accuracy of the radiation computation the user may increase the **Radiation resolution** (default value is 8), decrease the **Tolerance** (default value is $1e-3$) or increase the **Maximum number of adaptations** (default value is 3). Conversely changing these values in the opposite direction should decrease computational time. Also, higher values of the **Geometry shape order** under **Component** node may improve the results.

For an axisymmetric geometry, G_m and F_{amb} must be evaluated in a corresponding 3D geometry. The number of bounces of the rays in the azimuthal direction can be controlled by setting the **Azimuthal discretization** (default value is 100).

VIEW FACTOR

This section is available by clicking the **Show More Options** button () and selecting **Advanced Physics Options** in the **Show More Options** dialog.

Select the **Check consistency** checkbox to verify that the radiation directions are defined in a consistent manner in the model. A warning is issued in the solver when a boundary

or an external radiation source sees the back of a boundary emitting in the opposite direction. The boundary hit on its back has no radiative properties defined on the side receiving radiation. The following points should be considered when addressing this issue:

- The radiation directions of the boundaries and the opacity of the adjacent domains should be verified together with the variables `rad.Fbackside` and `rad.Fbacksideu`. These variables can be plotted to show the faces that see the opposite side of other boundaries.
- In the case of nonlinear geometry shape functions, this may also be due to the combination of a relatively coarse mesh with the **High order mesh** elements option selected in the interface settings.
- Conversely, quadrilateral mesh elements combined with curved boundaries can create inconsistencies in view factor calculations when **High-order mesh elements** is cleared.

Check consistency may require additional computational resources when using the hemicube algorithm. Once the consistency of the radiation model has been verified, it's recommended to clear this checkbox when using hemicube.

Select the **Store view factors on disk** checkbox to store the view factors (mutual and external irradiation) in the model after the next computation. View factors are reused automatically provided the mesh, the selection of the radiating boundaries, the radiation direction, external radiation sources or symmetry conditions are unchanged. However, adding or removing a feature or modifying a feature selection in the **Surface-to-Surface radiation** always forces to recompute view factors. The view factors are reused for example if only the value of the surface emissivity is changed. When coupled to a **Heat Transfer** interface through a **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling, changes for example in temperature, source, or flux conditions can be made in the **Heat Transfer** interface without having to recompute the view factors. The view factors are stored in the model file which may induce a noticeable increase in file size. When this checkbox is cleared and the model is saved, view factors are removed from the model file. View factors are also removed from the model when clearing the mesh and the solution. This option is not compatible with cluster computing.

Define the **View factors update threshold** to control the frequency of view factors computation in time-dependent simulation. This setting is used in models containing a **Moving Mesh** node under **Definitions**, when the **Symmetry for Surface-to-Surface**

Radiation feature uses one or more moving symmetry planes or with time-dependent specular surface properties.

- Keep the default value, **Every nonlinear iteration**, to update the view factors every nonlinear iteration. This option is the most computationally intensive.
- Select **Every Nth time** to update the view factors at least every N th time where N is the **View factors update time** (SI unit: s). For example, set **View factors update time** to $1\text{e-}3[\text{s}]$ to enforce the update every millisecond of simulation time.
- Select **Never update** to never update view factors. This option can be used when view factors update has little effect on the results (short timescales, small geometry changes). This option is the least computationally intensive.
- Select **User defined** to define the view factors update threshold manually. The view factors are updated each time the **Expression** changes by more than the **Tolerance**. By default, the tolerance is set to -1 to force the view factor to be recomputed every nonlinear iteration. For example, set **Expression** to `intop1(1)` and **Tolerance** to $0.15[\text{m}]$, where `intop1()` is an integration operator over a deforming boundary, to update view factors when the boundary length changes more than 0.15 m compared with the previous view factors calculation.

Select the **High order mesh elements** checkbox to use the mesh elements as they are defined by the geometry shape functions to compute view factors. This option is designed to improve the accuracy and is expected to be more computationally expensive. When a mesh element is concave with respect to the radiation direction, the self-irradiation is always accounted for when this option is selected. If not selected, the view factors are computed using linearized mesh elements.

JACOBIAN CONTRIBUTION

This section is available by clicking the **Show More Options** button () and selecting **Advanced Physics Options** in the **Show More Options** dialog.

Set the **Jacobian contribution**.

- Keep the default value, **Only local contributions to radiosity**, to exclude the irradiation contribution to the Jacobian matrix block corresponding to the radiosity degrees of freedom. This option is expected to reduce dramatically the time and the memory required to solve the model.
- Select **Include contributions from total irradiation** to include the irradiation in the Jacobian matrix and perform the full Jacobian evaluation. This option can ease the convergence.

Provided a converged solution with both options, the same solution is obtained within the solver tolerance.

When the Hemicube algorithm is selected in 2D and 3D, select **Lower integration order for irradiation** (the default) to reduce the number of evaluation of the irradiation. With this option enabled, the irradiation is evaluated:

- At the same order as the shape function order (radiosity or temperature when coupled to Heat Transfer) if the latter is even or equal to 1
- At the even integer following the shape function order otherwise.

By reducing the cost of the evaluation of the irradiation, this option also reduces the computational time and memory usage.

When this interface is coupled to Heat Transfer, this option can reduce the accuracy of energy balance evaluations.

DISCRETIZATION

Surface Radiosity

Select **Linear** (the default), **Quadratic**, **Cubic**, **Quartic**, or **Quintic** to define the discretization level used for the **Surface radiosity** shape function.

Fluence Rate

When a [Fluence Rate Calculation \(Surface-to-Surface Radiation Interface\)](#) domain condition is present, the fluence rate is stored in each mesh element of the selected domains, at Gauss points corresponding to the selected order. When the fluence rate is evaluated using E0 variable, an interpolation is performed from the values stored at Gauss points. To improve the accuracy of the E0 variable, the user may increase the Gauss points order by changing the **Fluence rate** value (default is 0). Conversely, changing this value in the opposite direction decreases computational time.



- [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 and Orbital Thermal Loads Interfaces).

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)
- [Specular Mirror](#) (Surface-to-Surface Radiation Interface)
- [Radiation Group](#) (Surface-to-Surface Radiation Interface)

Predefined Functions for the Surface-to-Surface Radiation Interface

When the **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, several predefined functions are available to obtain information about material properties. They can be used under the **Results** node to display the

wavelength dependency of the material properties, and the values actually used in the computation for each band. These predefined functions are available for the following surface material properties:

- Emissivity: `epsilon`
- Diffuse reflectivity: `rho_d`
- Specular reflectivity: `rho_s`
- Specular transmissivity: `tau_s`
- Critical angle: `theta_rad`

Several versions of the functions exist, averaged on the entire physics, averaged on a feature, or locally evaluated, and continuous, or piecewise constant. These functions start with an `f` and are followed by the symbol of the material property. Changing the scope and the suffix have different results:

- Physics averaged continuous functions are called with the scope of the physics and without suffix. They take one argument for the wavelength:
`rad.fepsilon(lambda)`
- Physics averaged piecewise constant functions are called with the scope of the physics and with the suffix “const”. They take one argument for the wavelength:
`rad.fepsilon_const(lambda)`
- Feature averaged continuous functions are called with the scope of the feature and without suffix. They take one argument for the wavelength:
`rad.dsrf1.fepsilon(lambda)`
- Feature averaged piecewise constant functions are called with the scope of the feature and with the suffix “const”. They take one argument for the wavelength:
`rad.dsrf1.fepsilon_const(lambda)`
- Locally evaluated continuous functions are called with the scope of the physics and with the suffix “at”. They take as many arguments as the dimension of the component to select the evaluation point, with the addition of the argument for the wavelength: `rad.fepsilon_at(x,y,z,lambda)`
- Locally evaluated piecewise constant functions are called with the scope of the physics and with the suffix “atconst”. They take as many arguments as the dimension of the component to select the evaluation point, with the addition of the argument for the wavelength: `rad.fepsilon_atconst(x,y,z,lambda)`

Feature averaged functions are available for the following list of boundary features:

- [Diffuse Mirror \(Surface-to-Surface Radiation Interface\)](#)

- Diffuse Surface (Surface-to-Surface Radiation Interface)
- Opaque Surface (Surface-to-Surface Radiation Interface)
- Semitransparent Surface (Surface-to-Surface Radiation Interface)

Physics averaged functions are averaged on all and only the features listed above.

Predefined plots are also available to add a built-in plot with the physics averaged functions for emissivity, diffuse reflectivity, specular reflectivity, and specular transmissivity.

The Orbital Thermal Loads Interface

The **Orbital Thermal Loads (otl)** interface (), found under the **Heat Transfer > Radiation** branch (), is used to model radiative loads on spacecraft, in particular radiation from the Sun and the Earth for satellites orbiting around Earth.

It defines the spacecraft radiative properties, orbit and orientation, orbital maneuvers, and planet properties. These inputs are used to compute and visualize the direct solar radiation, albedo, and planet radiative flux as well as the radiative heat transfer between the different spacecraft parts. It handles thermal radiation based on the radiosity method similarly to [The Surface-to-Surface Radiation Interface](#).

[Time Dependent](#), [Orbit Calculation](#), [Orbital Temperature](#) and [Orbit Thermal Loads](#) studies are supported in 3D.

In this section:

- [Settings for the Orbital Thermal Loads Interface](#)
- [Feature Nodes for the Orbital Thermal Loads Interface](#)
- [Predefined Functions for the Orbital Thermal Loads Interface](#)

	Theory for Orbital Thermal Loads
	When Planet properties are set to Earth, the default plots shows the spacecraft orbit around the Earth. The Earth is represented using an image, data:///physics/images/earth.jpg which credits come from NASA Goddard Space Flight Center Image by Reto Stöckli (land surface, shallow water, clouds). Enhancements by Robert Simmon (ocean color, compositing, 3D globes, animation). Data and technical support: MODIS Land Group; MODIS Science Data Support Team; MODIS Atmosphere Group; MODIS Ocean Group Additional data: USGS EROS Data Center (topography); USGS Terrestrial Remote Sensing Flagstaff Field Center (Antarctica); Defense Meteorological Satellite Program (city lights).

Settings for the Orbital Thermal Loads Interface

The settings are identical to the [Settings for the Surface-to-Surface Radiation Interface](#) with the difference that the **Transparent media refractive index** input is not available, the refractive index being assumed to be away equal to one.

Feature Nodes for the Orbital Thermal Loads Interface

This section details the nodes available with [The Orbital Thermal Loads Interface](#). All the nodes available in [The Surface-to-Surface Radiation Interface](#) are available:

- [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/>

In addition some global nodes specific to [The Orbital Thermal Loads Interface](#) are available: [Global Nodes for the Orbital Thermal Loads Interface](#).

GLOBAL NODES FOR THE ORBITAL THERMAL LOADS INTERFACE

[The Orbital Thermal Loads Interface](#) has the following global nodes:

- [Events Timeline](#)
- [Generate Events Interface](#)
- [Ground Pointing](#)
- [Implicit Event](#)
- [Orbital Parameters](#)
- [Planet Properties](#)
- [Spacecraft Axes](#)
- [Spacecraft Orientation](#)
- [Sun Properties](#)

Predefined Functions for the Orbital Thermal Loads Interface

The predefined functions available for the **Orbital Thermal Loads** interface are identical to the [Predefined Functions for the 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 does not 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](#)
- [Predefined Functions 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**.

	• <i>Radiative Heat Transfer in Finite Cylindrical Media</i>: Application Library path Heat_Transfer_Module/Verification_Examples/cylinder_participating_media • <i>Radiative Heat Transfer in a Utility Boiler</i>: 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. For integrations calculation, the upper bound of the spectral band for ambient radiation is predefined and set to 1 [mm] . This value can be modified by selecting the **Specify the upper bound for integrations** checkbox.
- Select **Multiple spectral bands** and set the value of the **Right endpoint** for each spectral band in the table, to define a spectral radiation model. The value of the **Right endpoint** must be entered without unit. Modify the **Wavelength unit** to set the unit of all the endpoints. **Right endpoint** values should be set in an ascending order. The

values of the **Left endpoint** for the next spectral band are updated automatically. It is possible to provide a definition of the surface properties for each spectral band.

	The first Left endpoint and the last Right endpoint are predefined and equal to 0 and $+\infty$, respectively. Parameters can be used to set the value of the Right endpoint.
	The wavelengths λ set in Solar and ambient and Multiple spectral bands are the wavelengths in vacuum.

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default for 2D and 3D components), or **P1 approximation** (the default for 2D axisymmetric components).

- When **Discrete ordinates method** is selected, [Opaque Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) and [Continuity on Interior Boundary \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) are automatically added as default features.
- When **P1 approximation** is selected, [Opaque Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) is automatically added as a default feature and both this and [Incident Intensity \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) are made available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or the context menu (all users). [Continuity on Interior Boundary \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) is not available.

	The choice of Radiation discretization method also offers different settings for the Participating Medium (Radiation in Participating Media Interface) (all methods), Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) (P1 approximation), and Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces) (P1 approximation) nodes.

Refractive Index

For either selection, define the **Refractive index** n_T (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 and 2D axisymmetric, 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 in 3D and $4N^2$ in 2D and 2D axisymmetric.



See [Discrete Ordinates Method \(DOM\)](#) for details about the S_N and T_N quadrature sets.

STABILIZATION

To display the stabilization section, click the **Show More Options** button () and select **Stabilization** in the **Show More Options** dialog.

When the **Radiation discretization method** is set to **Discrete ordinates method**, an alternative solver with a dedicated stabilization is proposed to replace the default solver

settings. By default, when solving for the discrete ordinates method equations, the radiative intensities are placed in segregated groups. This approach allows to decrease the requirements in terms of memory. When the **Use alternative solver with stabilization** checkbox is selected, no segregated group is created, a fully coupled approach is preferred. In order to avoid prohibitive memory requirements, an iterative solver is used. A stabilization term is added as well which can help for convergence. Note that this stabilization term acts as a diffusive term, and therefore impacts the result if it is too large.

The **Stabilization coefficient** controls the amount of stabilization added, the rule being that it should be as small as possible to reduce its impact on the results, and just enough to help the solver converge. This coefficient can be set to 0 to remove entirely the contribution of the stabilization but still use the alternative solver.



There is no established rule for the use and benefits of this alternative solver. Some cases may show dramatically reduced computation time for comparable memory requirements, and others may give similar solver behavior.

In order to use the alternative solver in an already existing study, click the **Reset Solver to Default** option in the **Solution** node or in the **Solver Configurations** node.

The **Performance Index** option has no influence on the alternative solver.

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)
- Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

Predefined Functions for the Radiation in Participating Media Interface

When the **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, several predefined functions are available to obtain information about material properties. They can be used under the **Results** node to display the wavelength dependency of the material properties, and the values actually used in the computation for each band. These predefined functions are available for the following domain material properties:

- Absorption coefficient: `kappaR`
- Scattering coefficient: `sigmaS`

Predefined functions are also available for the following surface material properties:

- Emissivity: `epsilon`

- Diffuse reflectivity: `rho_d`
- Diffuse transmissivity: `tau_d`
- Specular transmissivity: `tau_s`

Several versions of the functions exist, averaged on the entire physics, averaged on a feature, or locally evaluated, and continuous, or piecewise constant. These functions start with an `f` and are followed by the symbol of the material property. Changing the scope and the suffix have different results:

- Physics averaged continuous functions are called with the scope of the physics and without suffix. They take one argument for the wavelength:
`rpm.fepsilon(lambda)`
- Physics averaged piecewise constant functions are called with the scope of the physics and with the suffix “const”. They take one argument for the wavelength:
`rpm.fepsilon_const(lambda)`
- Feature averaged continuous functions are called with the scope of the feature and without suffix. They take one argument for the wavelength:
`rpm.os1.fepsilon(lambda)`
- Feature averaged piecewise constant functions are called with the scope of the feature and with the suffix “const”. They take one argument for the wavelength:
`rpm.os1.fepsilon_const(lambda)`
- Locally evaluated continuous functions are called with the scope of the physics and with the suffix “at”. They take as many arguments as the dimension of the component to select the evaluation point, with the addition of the argument for the wavelength: `rpm.fepsilon_at(x,y,z,lambda)`
- Locally evaluated piecewise constant functions are called with the scope of the physics and with the suffix “atconst”. They take as many arguments as the dimension of the component to select the evaluation point, with the addition of the argument for the wavelength: `rpm.fepsilon_atconst(x,y,z,lambda)`



The handling of predefined functions for domain material properties is identical.

Feature averaged functions are available for the [Participating Medium \(Radiation in Participating Media Interface\)](#) domain feature, and for the following list of boundary features:

- [Opaque Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#)
- [Semitransparent Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#)

Physics averaged functions are averaged on all and only the features listed above.

Predefined plots are also available to add a built-in plot with the physics averaged functions for domain and surface material properties separately.

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 nonemitting.

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](#)
- [Predefined Functions 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. For integrations calculation, the upper bound of the spectral band for ambient radiation is predefined and set to $1 [\text{mm}]$. This value can be modified by selecting the **Specify the upper bound for integrations** checkbox.
- Select **Multiple spectral bands** and set the value of the **Right endpoint** for each spectral band in the table, to define a spectral radiation model. The value of the **Right endpoint** must be entered without unit. Modify the **Wavelength unit** to set the unit of all the endpoints. **Right endpoint** values should be set in an ascending order. The values of the **Left endpoint** for the next spectral band are updated automatically. It is possible to provide a definition of the surface properties for each spectral band.



The first **Left endpoint** and the last **Right endpoint** are predefined and equal to 0 and $+\infty$, respectively.

Parameters can be used to set the value of the **Right endpoint**.



The wavelengths λ set in **Solar and ambient** and **Multiple spectral bands** are the wavelengths in vacuum.

Radiation Discretization Method

Select a **Radiation discretization method**: **Discrete ordinates method** (the default for 3D and 2D components), or **PI approximation** (the default for 2D axisymmetric components).

- 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_T (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 and 2D axisymmetric, 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 in 3D and $4N^2$ in 2D and 2D axisymmetric.



See [Discrete Ordinates Method \(DOM\)](#) for details about the S_N and T_N quadrature sets.

STABILIZATION

To display the stabilization section, click the **Show More Options** button () and select **Stabilization** in the **Show More Options** dialog.

When the **Radiation discretization method** is set to **Discrete ordinates method**, an alternative solver with a dedicated stabilization is proposed to replace the default solver settings. By default, when solving for the discrete ordinates method equations, the radiative intensities are placed in segregated groups. This approach allows to decrease the requirements in terms of memory. When the **Use alternative solver with stabilization** checkbox is selected, no segregated group is created, a fully coupled approach is

preferred. In order to avoid prohibitive memory requirements, an iterative solver is used. A stabilization term is added as well which can help for convergence. Note that this stabilization term acts as a diffusive term, and therefore impacts the result if it is too large.

The **Stabilization coefficient** controls the amount of stabilization added, the rule being that it should be as small as possible to reduce its impact on the results, and just enough to help the solver converge. This coefficient can be set to 0 to remove entirely the contribution of the stabilization but still use the alternative solver.



There is no established rule for the use and benefits of this alternative solver. Some cases may show dramatically reduced computation time for comparable memory requirements, and others may give similar solver behavior.

In order to use the alternative solver in an already existing study, click the **Reset Solver to Default** option in the **Solution** node or in the **Solver Configurations** node.

The **Performance Index** option has no influence on the alternative solver.

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)
- Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

Predefined Functions for the Radiation in Absorbing-Scattering Media Interface

The predefined functions available for the **Radiation in Absorbing-Scattering Media** interface are identical to the [Predefined Functions for the Radiation in Participating Media Interface](#) with the exception that the predefined functions for domain material properties are available for the [Absorbing-Scattering Medium \(Radiation in Absorbing-Scattering Media Interface\)](#) feature instead of the [Participating Medium \(Radiation in Participating Media Interface\)](#) feature.

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** checkbox 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** checkbox is also selected by default and is only available when the **Streamline diffusion** checkbox is selected.

INCONSISTENT STABILIZATION

This section is available by clicking the **Show More Options** button () and selecting **Stabilization** in the **Show More Options** dialog. The **Isotropic diffusion** checkbox 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 Equilibrium Moisture Transport in Porous Media Interface

The **Equilibrium 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. A [Hygroscopic Porous Medium](#) model is active by default on all domains, with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**.

The water conservation equation defined in porous media domains corresponds to the convection-diffusion equation for the total moisture content. This equation is valid when vapor phase and liquid phase are in equilibrium. The relation between the relative humidity and the moisture content is described by the sorption isotherm curve. If not, use [The Nonequilibrium Moisture Transport in Porous Media Interface](#) instead.

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 **Equilibrium 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 in Porous Media

The Nonequilibrium Moisture Transport in Porous Media Interface

The **Nonequilibrium 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. A [Hygroscopic Porous Medium](#) model is active by default on all domains, with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation**.

The water mass conservation equations in the liquid and in the moist air phases of the porous medium are solved and coupled through a transfer term proportional to the difference between the partial vapor pressure and the equilibrium vapor pressure. If this difference can be neglected, use [The Equilibrium Moisture Transport in Porous Media Interface](#) instead.

When this physics interface is added, these default nodes are added to the **Model Builder: Hygroscopic Porous Medium, Insulation** (the default boundary condition), overridden by **Nonequilibrium Boundary** (showing all the boundaries adjacent to domains where both the relative humidity and liquid saturation are solved for the moist air and liquid phases), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Nonequilibrium Moisture Transport in Porous Media** to select physics features from the context menu.

Specific subnodes are also present by default under the **Liquid Water** and **Moist Air** subnodes of the **Hygroscopic Porous Medium** node:

- **Initial Values**, which allows to set a specific initial liquid water saturation and relative humidity.
- **Insulation**, which sets a no flux condition for each water phase.

Other subnodes implementing boundary conditions specific to the liquid and moist air phases can be added to model flow conditions and fluxes. To do so, right-click a **Liquid Water** or **Moist Air** node to select physics features from the context menu.

Liquid Water Flux and Vapor Flux

In the liquid water flux and vapor flux features available under the **Liquid Water** and **Moist Air** subnodes, the user input corresponds to the moisture flux per total unit surface. It is multiplied by the volume fraction of each phase and added into the

corresponding water mass conservation equation. The surface fraction is approximated by the volume fraction.

By default, the shape functions used for the relative humidity and the liquid saturation 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 in Porous Media

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 <i>COMSOL Multiphysics Reference Manual</i> 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 Moisture Transport in Free and Porous Media Interface

The **Moisture Transport in Free and Porous Media (mt)** interface () is used to model moisture transport in moist air and into the pores of a porous medium, through convection and diffusion of vapor in moist air, and convection and capillary flow of liquid water. Both a **Moist Air (Moisture Transport Interface)** model and a **Hygroscopic Porous Medium** model are added by default. The first one is active on all domains, while the second one has no selection. Also, the **Equilibrium between liquid and gas phases** option of the **Hygroscopic Porous Medium** feature is set to **Nonequilibrium formulation**.

When this physics interface is added, these default nodes are added to the **Model Builder**: **Moist Air**, **Hygroscopic Porous Medium**, **Insulation** (the default boundary condition), **Nonequilibrium Boundary** (showing all the boundaries adjacent to domains where both the relative humidity and liquid saturation are solved for the moist air and liquid phases), **Porous Interface**, 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 Free and Porous Media** to select physics features from the context menu.

Specific subnodes are present by default under the **Liquid Water** and **Moist Air** subnodes of the **Hygroscopic Porous Medium** node:

- **Initial Values**, which allows to set a specific initial liquid water saturation and relative humidity.
- **Insulation**, which sets a no flux condition for each water phase.

Other subnodes implementing boundary conditions specific to the liquid and moist air phases can be added to model flow conditions and fluxes. To do so, right-click a **Liquid Water** or **Moist Air** node to select physics features from the context menu.

Specific subnodes are also present under the **Porous Interface** node:

- **Free-Porous Interface**, which allows to set the interface condition between free domains (**Moist Air**) and porous domains (**Hygroscopic Porous Medium**, **Building Material**).
- **Porous-Porous Interface**, which allows to set the interface condition between two porous domains.

Liquid Water Flux and Vapor Flux

In the liquid water flux and vapor flux features available under the **Liquid Water** and **Moist Air** subnodes, the user input corresponds to the moisture flux per total unit surface. It is multiplied by the volume fraction of each phase and added into the corresponding water mass conservation equation. The surface fraction is approximated by the volume fraction.

By default, the shape functions used for the relative humidity and the liquid saturation in all domains 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 Solids Interface

The **Moisture Transport in Solids (mts)** interface () is used to model moisture transport in the pores of a porous medium, through convection and diffusion of vapor in moist air, and convection and capillary flow of liquid water. A [Porous Medium](#) model is active by default on all domains.

The water conservation equation defined in porous media domains corresponds to the convection-diffusion equation for the total moisture content. This equation is valid when the amount of water in vapor phase and in liquid phase are in equilibrium, which corresponds to the saturation conditions. An additional equation is solved for the transport of dry air to account for changes in the moist air pressure.

When this physics interface is added, these default nodes are added to the **Model Builder: Porous Medium, No Flux** (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 Solids** to select physics features from the context menu.

By default, the shape functions used for the relative humidity and the moist air pressure are **Quadratic**.



See [The Moisture Transport in Solids Interface](#) in the *Porous Media Flow Module User's Guide* for details.



[Theory for Moisture Transport in Solids](#)

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:

- Battery Layers
- Bioheat
- Biological Tissue
- Building Material
- Cell Periodicity
- Convectively Enhanced Conductivity
- Cross Section
- Fluid
- Fluid (Porous Medium)
- Geothermal Heating
- Heat Source
- Heat Source (Pellets)
- Immobile Fluids (Porous Medium)
- Initial Values
- Initial Values (Pellets)
- 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
- Optically Thick Participating Medium
- Out-of-Plane Heat Flux
- Out-of-Plane Radiation
- Pellets (Porous Medium)
- Pellet-Fluid Interface (Porous Medium)
- Phase Change Material
- Porous Matrix (Porous Medium, Moist Porous Medium)
- Porous Medium
- Pressure Work
- Shape Memory Alloy
- Solid
- Solid with Translational Motion
- Thermal Damage
- Thermal Dispersion
- Thermoelastic Damping
- Thickness
- Translational Motion
- Viscous Dissipation

The following domain nodes are available for the Surface-to-Surface Radiation, Radiation in Participating Media, Radiation in Absorbing-Scattering Media, and Radiative Beam in Absorbing Media interfaces:

- Absorbing Medium (Radiative Beam in Absorbing Medium Interface)
- Absorbing-Scattering Medium (Radiation in Absorbing-Scattering Media Interface)
- Fluence Rate Calculation (Surface-to-Surface Radiation Interface)
- Initial Values (Radiative Beam in Absorbing Medium Interface)
- Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Opacity (Surface-to-Surface Radiation and Orbital Thermal Loads Interfaces)
- Participating Medium (Radiation in Participating Media Interface)
- Radiative Source



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 nonemitting medium.

It calculates the radiative heat source term Q_r (SI unit: W/m^3), defined by:

$$Q_r = \kappa G$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2), defined by

$$G = \int_{4\pi} I(\Omega) d\Omega$$

where

- $I(\Omega)$ is the radiative intensity (SI unit: $W/(m^2 \cdot sr)$) at a given position following the Ω direction, that satisfies the *radiative transfer equation* with no emission term

$$\Omega \cdot \nabla I(\Omega) = -\beta I(\Omega) + \frac{\sigma_s}{4\pi} \int_0^{4\pi} I(\Omega') \phi(\Omega', \Omega) d\Omega'$$

- $\beta = \kappa + \sigma_s$ is the extinction coefficient (SI unit: $1/m$).
- σ_s is the scattering coefficient (SI unit: $1/m$).
- $\phi(\Omega', \Omega)$ is the scattering phase function (dimensionless)

If the **Radiation discretization method** is **Discrete ordinates method**, G is defined by

$$G = \sum_{i=1}^N \omega_i I_i$$

and, in a cartesian coordinates system

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^N \omega_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \quad (6-1)$$

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 j^{th} quadrature weight.

If the **Radiation discretization method** is **P1 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	P1
Optical thickness validity	All	$\tau \gg 1$
Absorption modeling	Yes	Yes
Scattering modeling	Linear Polynomial (in 2D and 3D)	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 **Heney-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 **Heney-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** checkbox to define a phase function such as

$$\int_{4\pi} \phi(\mu_0) d\Omega = 4\pi$$

The normalization is automatically applied for the **Henye-Greenstein** option.

For **Linear anisotropic**, **Polynomial anisotropic**, and **Henye-Greenstein**, select the **Wavelength-dependent scattering type** checkbox 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

Battery Layers

The **Battery Layers** domain node, available in 3D, allows for modeling of heat transfer in battery layers cells using a homogenized approach, where the individual layers of the cell do not need to be resolved in the computational mesh.

The node is available with a Battery Design Module license only.

The effective **In-layer** (k_{il}) and **Through-layer thermal conductivities** (k_{tl}), **Density** and **Heat capacity** values are specified by the user.

The homogenization of the heat equation (for solids) is performed by the use of anisotropic thermal conductivity tensors, which are defined based on the configuration (cylindrical or pouch) of the battery layers, and the effective thermal conductivity in the in-layer and through-layer directions.

The **Layer configuration** setting specifies how the conductivity tensor should be calculated, and what coordinate system should be used for defining the heat equation. If the layers are oriented in a **Flat parallel (pouch)** configuration, the thermal conductivity tensor is defined using the default global cartesian coordinates (x, y, z). This would be the typical configuration for a pouch cell.

The **Through-layer direction** (plane normal) may be set to coincide with either the x, y , or z coordinate axis, and the tensor is then defined in the (x, y, z) system as either

$$k = \begin{bmatrix} k_{tl} & 0 & 0 \\ 0 & k_{il} & 0 \\ 0 & 0 & k_{il} \end{bmatrix},$$

$$k = \begin{bmatrix} k_{il} & 0 & 0 \\ 0 & k_{tl} & 0 \\ 0 & 0 & k_{il} \end{bmatrix},$$

or

$$k = \begin{bmatrix} k_{il} & 0 & 0 \\ 0 & k_{il} & 0 \\ 0 & 0 & k_{tl} \end{bmatrix},$$

respectively.

For cylindrical batteries, the layers are typically wound around a cylindrical axis, and may be approximated by defining the anisotropic transport properties in a cylindrical coordinate system.

For a **Spirally wound (cylindrical)** configuration, the thermal conductivity tensor is defined using the cylindrical coordinates (r_c, ϕ_c, z_c) with the origin placed in the center of each battery cylinder, and the z_c -axis pointing along the **Longitudinal axis** of the cylinders, which is specified by the user to coincide with either the x -, y -, or z -axis.

The thermal conductivity tensor then defined in the (r_c, ϕ_c, z_c) system as

$$k = \begin{bmatrix} k_{tl} & 0 & 0 \\ 0 & k_{il} & 0 \\ 0 & 0 & k_{il} \end{bmatrix}.$$

COORDINATE SYSTEM SELECTION

The settings of this section are locked and cannot be altered by the user.

Multicylindrical coordinate systems, supporting multiple disjoint domains, are created automatically when **Spirally wound (cylindrical)** is selected as **Layer configuration** in the **Battery Layers** section.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Specific Media > Battery Layers

Heat Transfer in Solids > Specific Media > Battery Layers

Heat Transfer in Fluids > Specific Media > Battery Layers

Heat Transfer in Porous Media > Specific Media > Battery Layers

Ribbon

Physics tab with **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, or **Heat Transfer in Porous Media** selected:

Domains > Specific Media > Battery Layers

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 (m³/s)/m³), describes the volume of blood per second that flows through a unit volume of tissue.
- **Density, blood** ρ_b , which is the mass per volume of blood.
- **Metabolic heat source** Q_{met} , which describes heat generation from metabolism. Enter this quantity as the unit power per unit volume.



- [Theory for Bioheat Transfer](#)
- [Biological Tissue](#)
- [Thermal Damage](#)



Hepatic Tumor Ablation: Application Library path
Heat_Transfer_Module/Medical_Technology/tumor_ablation

Microwave Heating of a Cancer Tumor: Application Library path
Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy

Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis:
Application Library path **Heat_Transfer_Module/Medical_Technology/conical_dielectric_probe**

LOCATION IN USER INTERFACE

A default **Bioheat** node is automatically added to the [Biological Tissue](#) node.

Context Menus

Bioheat Transfer > Biological Tissue > Bioheat

Ribbon

Physics tab with **Biological Tissue** selected in the model tree:

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.

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics interface for example), you can select on which frame the thermal conductivity is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables $ht.k_{xx}$, $ht.k_{yy}$, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables $ht.k_{xx}$, $ht.k_{yy}$, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{(x,y,z)} = \mathbf{R}k_{(X,Y,Z)}\mathbf{R}^T$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

THERMODYNAMICS, SOLID

The default **Density** ρ and **Heat capacity at constant pressure** C_p are taken **From material**. See [Material Density in Features Defined in the Material Frame](#) if a temperature-dependent density should be set.

The heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit mass.

For **User defined** enter other values or expressions.



- [Theory for Bioheat Transfer](#)
- [Bioheat](#)
- [Thermal Damage](#)

	There are specific predefined materials available in the Bioheat material database. See Materials Overview and Bioheat Materials Library in the <i>COMSOL Multiphysics Reference Manual</i>.
	<i>Hepatic Tumor Ablation</i>: Application Library path Heat_Transfer_Module/Medical_Technology/tumor_ablation <i>Microwave Heating of a Cancer Tumor</i>: Application Library path Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy <i>Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis</i>: 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: J/(m³·K)) is the effective volumetric heat capacity at constant pressure.
- k_{eff} (SI unit: W/(m·K)) is the effective thermal conductivity (a scalar or tensor if the thermal conductivity is anisotropic).
- L_v (SI unit: J/kg) is the latent heat of evaporation.
- δ_p (SI unit: s) is the vapor permeability tensor.
- ϕ_w (dimensionless) is the relative humidity.
- p_{sat} (SI unit: Pa) is the vapor saturation pressure.
- 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, 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.

Relative Humidity

This section contains an input for the definition of the relative humidity, which is used in the right hand side of [Equation 6-4](#).

The default **Relative humidity** ϕ_w is **User defined**. When additional physics interfaces are added to the model, the relative humidity variables defined by these physics interfaces can also be selected from the list. For example, if a **Moisture Transport** interface is added, you can select **Relative humidity (mt/pm1)** from the list. The **Common model input** option is the variable `minput.phi`, set to 0 by default. To edit it, click the **Go to Source** button (), 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:

$$k_{\text{eff}} = k_s \left(1 + \frac{bw(\phi_w)}{\rho_s} \right)$$

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/m³) is the density of the dry solid.
- $C_{p,s}$ (SI unit: J/(kg·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**, select **Isotropic**, **Diagonal**, or **Symmetric** based on the vapor permeability characteristics, and enter values or expressions for the tensor components.
- **Vapor resistance factor** μ to define the vapor permeability δ_p as

$$\delta_p = \mu^{-1}\delta$$

where δ (SI unit: s) is the vapor permeability of still air. The default **Vapor resistance factor** is taken **From material**. For **User defined**, enter another value or expression.

If the node was added automatically after selecting the predefined multiphysics interface **Heat and Moisture Transport**, the building material properties of the multiphysics node **Heat and Moisture** are used by default and the inputs are not editable. To edit these fields, click **Make All Model Inputs Editable** () in the **Model Inputs** section.

	• The Heat Transfer in Building Materials Interface • The Heat and Moisture Transport 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

Cell Periodicity

Use the **Cell Periodicity** node to model a repeating unit cell (RUC) or representative volume element (RVE) of a larger repetitive structure. Periodic or uniform temperature boundary conditions are then used on the cell outer boundaries.

From the **Cell Periodicity** node, it is possible to compute the thermal conductivity tensor, the heat capacity at constant pressure and the density for the equivalent homogenized material.

This node is available for 3D and 2D components. Since the homogenization is only for heat transfer by conduction, the **Cell Periodicity** node is only applicable on [Solid](#) domain feature.

CELL PROPERTIES

Select the **Boundary condition type** among **Periodic temperature** (default) and **Uniform temperature gradient**.

- When **Periodic temperature** is selected, the temperature is periodic over the cell outer boundaries. **Boundary Pair** subfeature must then be applied to all the cell outer boundaries.
- When **Uniform temperature gradient** is selected, the temperature gradient is uniform over the cell outer boundaries. This option can be used to prescribe boundaries when the cell geometry does not represent a periodic microstructure.

The **Temperature scale factor** (dimensionless, default value 1) multiplied by 1 kelvin defines the amplitude of temperature used for the calculation of equivalent homogenized material properties. Changing this value only affects results if the material properties used in the unit cell depend on the temperature.



The temperature over the cell outer boundaries is set to the **Reference temperature** defined in the **Physical Model** section of the physics interface, plus the amplitude of temperature defined by the **Temperature scale factor**.

Load Group, Material and Study Generation

The equivalent material properties for a homogenized material can be created, and made accessible as a material to be used in other components. In order to do this, a number of fundamental load cases must be analyzed. The necessary definitions and steps required for such an analysis can be automated as described below.

On the **Cell Properties** section toolbar, the **Automated Model Setup** () menu has the following entries:

- **Create Load Groups and Study** (). This option can be selected when the **Domain Selection** of the **Cell Periodicity** feature is not empty. When selected, the following changes will be made to the model:
 - A number of load groups will be created under **Global Definitions**. They are collected in a group named **Load Groups for Heat Transfer Cell Periodicity**. The load groups correspond to unit loads along different axes.
 - A new study, named **Cell Periodicity Study htcp1**, with **ht** and **cp1** referring respectively to the tags of the parent physics interface and the corresponding **Cell Periodicity** feature, will be created. This study contains a stationary study step that

solves for the parent physics interface only. In the **Study Extensions** section of the new **Cell Periodicity Study**, one load case is added for each load group.

- **Create Material by Reference** (). This option can be selected when the **Domain Selection** of the **Cell Periodicity** feature is not empty. When you select it, a new material will be created under **Global > Materials**. It contains the thermal conductivity tensor, the heat capacity at constant pressure, and the density of the equivalent homogenized material. The name of this material is **Homogeneous Material (htcpImat)**, with **ht** and **cpI** referring respectively to the tags of the parent physics interface and the corresponding **Cell Periodicity** feature.
- **Create Material by Value** (). This option can be selected when results from the study generated by **Create Load Groups and Study** are available. This option is similar to **Create Material by Reference**, with the difference that the material properties are numerical values, and not references to variables in the node. The benefits of this option is faster computation when the material is used by other physics interfaces. Also, if the **Cell Periodicity** node is deleted, the homogenized material generated from it can still be used in subsequent computations.



For this type of analysis to work correctly, it is important not to manually edit the generated nodes. By clicking the **Create Load Groups and Study** () button again, all settings in the generated nodes are reset to their default values.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default), **Weak constraints** or **Nitsche constraints**.



- [Constraint Reaction Terms](#)
- [Weak Constraints](#)
- [Constraint Settings](#)

	• Theory for Cell Periodicity • Using Load Cases and Materials in the <i>COMSOL Multiphysics Reference Manual</i>.
	<i>Equivalent Properties of Periodic Microstructures</i> : Application Library path Heat_Transfer_Module/Applications/equivalent_properties_of_periodic_microstructures

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Specific Media > Cell Periodicity

Heat Transfer in Solids > Specific Media > Cell Periodicity

Heat Transfer in Fluids > Specific Media > Cell Periodicity

Heat Transfer in Porous Media > Specific Media > Cell Periodicity

Heat Transfer in Building Materials > Specific Media > Cell Periodicity

Bioheat Transfer > Specific Media > Cell Periodicity

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids**, or any version of the Heat Transfer interface selected:

Domains > interface > Specific Media > Cell Periodicity

Convectively Enhanced Conductivity

This subnode takes into account convective heat flux by enhancing the fluid thermal conductivity according to the Nusselt number. When added under the **Fluid** or the **Moist Air** node, it sets the fluid velocity to zero, and the user-defined or predefined velocity model input is ignored.

CONVECTIVELY ENHANCED CONDUCTIVITY

The following options are available in the **Nusselt number correlation** list:

- **Horizontal cavity heated from below**, for which values for the **Cavity height** H and the **Temperature difference** ΔT should be specified for the computation of the Nusselt number. Unfold the **Sketch** section for more details on the required parameters.
- **Vertical rectangular cavity**, for which values for the **Cavity height** H , the **Plate distance** L , and the **Temperature difference** ΔT should be specified for the computation of the

Nusselt number. Unfold the **Sketch** section for more details on the required parameters.

- **User defined**, for which a value for Nu should be specified directly.

For the two first options, select **Automatic** (default) or **User defined** to define the **Temperature difference** ΔT . When **Automatic** is selected the temperature difference is evaluated as the difference between the maximal and the minimal temperature on the exterior boundaries of the feature selection.

DYNAMIC VISCOSITY

This section is not available when the **Convectively Enhanced Conductivity** feature is added under the **Fluid** node with **Fluid type** set as **Moist air**, or under the **Moist Air** node. Else, the **Dynamic viscosity** μ used to calculate the Nusselt number should be set.

	When the Fluid type is Moist air , or when the Convectively Enhanced Conductivity feature is added under the Moist Air node, the natural convection due to vapor concentration gradients is neglected. Indeed, only thermally induced variations of the density are accounted for in these correlations.
	When the Nonisothermal Flow multiphysics coupling node is added, the Convectively Enhanced Conductivity feature is overridden by the multiphysics coupling node's contribution, and convection is explicitly calculated by using the predefined velocity model input.
	<i>Evaporative Cooling of Water</i> : Application Library path Heat_Transfer_Module/Phase_Change/evaporative_cooling
	Equivalent Thermal Conductivity Correlations

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Fluids > Fluid > Convectively Enhanced Conductivity**
- Heat Transfer in Moist Air > Moist Air > Convectively Enhanced Conductivity**

More locations are available. For example:

Heat Transfer in Solids > Fluid > Convectively Enhanced Conductivity

Ribbon

Physics tab with **Fluid** or **Moist Air** selected in the model tree:

Attributes > Convectively Enhanced Conductivity

Cross Section

Use this node with 1D components to model domains with another cross sectional area or another cross sectional perimeter than the global one that is used in the Heat Transfer interface **Physical Model** section. In 1D geometries, the temperature is assumed to be constant in the radial direction, and the heat equation is modified to take that into account. See [Equation 4-181](#) and [Equation 4-182](#).

CROSS SECTION

Enter values for the **Cross sectional area** A_c and the **Cross sectional perimeter** P_c to set the cross section of the domain in the plane perpendicular to the 1D geometry.



The **Cross Section** feature is disabled when at least one of the multiphysics couplings ([Heat Transfer with Surface-to-Surface Radiation](#), [Heat Transfer with Radiation in Participating Media](#), [Heat Transfer with Radiative Beam in Absorbing Media](#)) is active while solving on the same domain. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.



[Out-of-Plane Heat Transfer](#)

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Cross Section

Heat Transfer in Solids > Cross Section

Heat Transfer in Fluids > Cross Section

Heat Transfer in Porous Media > Cross Section

Heat Transfer in Building Materials > Cross Section
Bioheat Transfer > Cross Section

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, or **Bioheat Transfer** selected:

Domains > interface > Cross Section

Fluence Rate Calculation (Surface-to-Surface Radiation Interface)

The **Fluence Rate Calculation** node is used to compute the fluence rate within selected domains. Fluence rate is defined as the total radiant power that would hit a small sphere located within the domain, divided by the cross sectional area of the sphere. The sphere is assumed to be much smaller than any geometric details but much larger than the wavelength of the radiation.

The variable for fluence rate is `<phys>.E0`, where `<phys>` is the name of the **Surface-to-Surface Radiation** interface, for example `rad.E0`. When several spectral bands are defined, the fluence rate can be evaluated per wavelength with `<phys>.E0_band<k>`, where `<k>` is the band number. This variable is precomputed and its values are stored at Gauss points on the mesh, which makes it very fast to evaluate. The Gauss points order is defined in the discretization section of the physics interface node.

Another variable, `<phys>.fr_band<k>`, can also be used to visualize the fluence rate. The difference is that this variable is evaluated using an operator at the exact position where it is requested (instead of interpolating from the Gauss point values). As these values are not cached, the evaluation time is much more important compared to `<phys>.E0`.

See [View Factor Evaluation](#) for more details on the underlying theory.

LOCATION IN USER INTERFACE

Context Menus

Surface-to-Surface Radiation > Fluence Rate Calculation

Ribbon

Physics tab with **Surface-to-Surface Radiation** selected in the model tree:

Domains > Fluence Rate Calculation

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/m^3) is the heat source (or sink). Add one or more heat sources as separate physics features. See the [Heat Source](#) node and the [Viscous Dissipation](#) and [Pressure Work](#) subnodes, for example.

For a steady-state problem the temperature does not change with time, and the first term disappears.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include enthalpy (the energy flux, for example).

It is also used if the ideal gas law is applied. See [Thermodynamics, Fluid](#).

The default **Absolute pressure** p_A is taken from **Common model input**. It corresponds to the `minput.pA` variable, set to 1 atm by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section. When additional physics interfaces are added to the model, the absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Laminar Flow** interface is added you can select **Absolute pressure (spf)** from the list. The last option is **User defined**.

HEAT CONVECTION

The default **Velocity field** $\mathbf{u}$ 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 ideal gas law when **Ideal gas** is selected.

When the density is not taken from a **Nonisothermal Flow** or a **Nonisothermal Mixture Model** coupling node, you should select a **Fluid type** option for the specification of the material properties. The available **Fluid type** options are **From material**, **Ideal gas**, and **Gas/Liquid** (default). After selecting a **Fluid type** from the list, further settings display underneath. See [Nonisothermal Mixture Model](#) in the *CFD Module User's Guide* for details.

From Material

This option automatically detects whether the material applied on each domain selection is an ideal gas or not, and uses the relevant properties from the **Material** node for either case.

By using the following definition of the density for an ideal gas:

$$\rho = \frac{p_A}{R_s T}$$

with R_s the specific gas constant, p_A the absolute pressure, and T the temperature, the evaluation of the isobaric compressibility coefficient, α_p , and of the isothermal Joule-Thomson coefficient, μ_{JT} , is simplified. This may improve efficiency, when computing pressure work in compressible nonisothermal flows for example, or when modeling inflow conditions.



The Air material, from the Built-in materials database, has an **Ideal gas** property group, and is thus detected as an ideal gas by the **Fluid** node. The [Liquids and Gases Material Library](#), available with some COMSOL products, also provides such materials.

The **Density** and the **Heat capacity at constant pressure** are automatically taken **From material** and no further setting is required.

Automatic calculation of the ratio of specific heats

The **Ratio of specific heats** is calculated automatically using Mayer's relation:

$$\gamma = \frac{\rho C_p \chi_t}{\rho C_p \chi_t - T \alpha_p^2}$$

considering that:

- α_p is the coefficient of thermal expansion (SI unit: 1/K):

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

- χ_t is the isothermal compressibility (SI unit: 1/Pa):

$$\chi_t = -\frac{1}{\rho} \frac{\partial \rho}{\partial p}$$

For the specification of user defined material properties, the [Ideal Gas](#) or [Gas/Liquid](#) options should be used instead.

Ideal Gas

This option uses the ideal gas law to describe the fluid. Only two properties are needed to define the thermodynamics of the fluid:

- The gas constant, with two options for the **Gas constant type: Specific gas constant** R_s or **Mean molar mass** M_n . If **Mean molar mass** is selected the software uses the universal gas constant $R = 8.314 \text{ J}/(\text{mol}\cdot\text{K})$, 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. By default, the **Ratio of specific heats** is set to Automatic, see [Automatic calculation of the ratio of specific heats](#).



- [Theory for Heat Transfer in Porous Media](#)
 - [Theory for Heat Transfer in Fluids](#)
-

	- When the density is taken from a Nonisothermal Flow coupling node, the Heat capacity at constant pressure remain to be specified for a general gas or liquid. Select Ideal gas: ratio of specific heats or Ideal gas: heat capacity at constant pressure to specify only one property in the case of an ideal gas. - When the Heat Transfer interface is coupled to a Phase Transport interface and to a Fluid Flow interface via the Nonisothermal Flow and Nonisothermal Mixture Model multiphysics couplings, all the material properties are taken from the coupling nodes. See Nonisothermal Flow and Nonisothermal Mixture Model in the <i>CFD Module User's Guide</i> for details.
	The Viscous Dissipation (for heat generated by viscous friction), Pressure Work (available with some COMSOL products), and Convectively Enhanced Conductivity subnodes are available from the context menu (right-click the parent node) or from the Physics toolbar, Attributes menu.
	<i>Heat Sink</i>: Application Library path Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/heat_sink
	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Fluid

Heat Transfer in Solids > Fluid

Heat Transfer in Fluids > Fluid

Heat Transfer in Porous Media > Fluid

Heat Transfer in Building Materials > Fluid

Bioheat Transfer > Fluid

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 the heat transfer equation of the [Porous Medium](#) parent node, to model heat transfer in a porous matrix, possibly consisting of several solids, and filled with a mobile fluid, and one or more immobile fluids.

The fluid can be specified as a general gas or liquid, as an ideal gas, or as moist air.



- The total volume fraction and material properties of immobile fluids can be specified in the [Immobile Fluids \(Porous Medium\)](#) subnode.
- The porosity and material properties of the solid matrix can be specified in the [Porous Matrix \(Porous Medium, Moist Porous Medium\)](#) subnode.
- When the porous medium is a packed bed of pellets, the material properties of the pellets can be specified in the [Pellets \(Porous Medium\)](#) subnode.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.



This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include enthalpy (the energy flux, for example).

It is also used if the ideal gas law is applied. See [Thermodynamics, Fluid](#).

The default **Absolute pressure** p_A is taken from **Common model input**. It corresponds to the variable `minput.pA`, set to 1 atm by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section. When additional physics interfaces are added to the model, the absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Laminar Flow** interface is added, you can select **Absolute pressure (spf)** from the list. The last option is **User defined**.

Concentration

This section can be edited whenever a material property is dependent on concentration, for example, when the **Fluid type** is set to **Moist air** with the **Input quantity** set to **Concentration**.

In the **Concentration** c (SI unit: mol/m^3 or kg/m^3) list, select an existing concentration variable from another physics interface, if there are concentration

variables, **User defined** to enter a value or expression for the concentration, or **Common model input** that corresponds to the `minput.c` variable.

HEAT CONVECTION

The default **Velocity field $\mathbf{u}$** 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.

Note that this section is not available when **Porous medium type** is set to **Local thermal equilibrium** and **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list of the **Porous medium** parent node.

THERMODYNAMICS, FLUID

This section defines the thermodynamics properties of the fluid.

The heat capacity at constant pressure $C_{p,f}$ describes the amount of thermal energy required to produce a unit temperature change in a unit mass.

The ratio of specific heats γ is the ratio between the heat capacity at constant pressure, $C_{p,f}$, and the heat capacity at constant volume, $C_{v,f}$. When using the ideal gas law to describe a fluid, it is sufficient to specify γ to evaluate $C_{p,f}$. For common diatomic gases such as air, $\gamma = 1.4$ is the standard value. Most liquids have $\gamma = 1.1$ while water has $\gamma = 1.0$. γ is used in the streamline stabilization and in the ideal gas law when **Ideal gas** is selected.

The ratio of specific heats can be calculated automatically when set to **Automatic** by using Mayer's relation:

$$\gamma = \frac{\rho C_p \chi_t}{\rho C_p \chi_t - T \alpha_p^2}$$

considering that

- α_p is the coefficient of thermal expansion (SI unit: 1/K):

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

- χ_t is the isothermal compressibility (SI unit: 1/Pa):

$$\chi_t = -\frac{1}{\rho} \frac{\partial \rho}{\partial p}$$

The available options for the **Fluid type** are **Gas/Liquid** (default), **Moist air**, or **Ideal gas**. After selecting a **Fluid type** from the list, other settings are displayed below.

Gas/Liquid

This option specifies the **Density**, the **Heat capacity at constant pressure**, and the **Ratio of specific heats** for a general gas or liquid.

Ideal Gas

This option uses the ideal gas law to describe the fluid. Only two properties are necessary to define the thermodynamics of the fluid:

- The gas constant, with two options for the **Gas constant type**: **Specific gas constant** R_s or **Mean molar mass** M_n . If **Mean molar mass** is selected, the software uses the universal gas constant $R = 8.314 \text{ J}/(\text{mol}\cdot\text{K})$, 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** to define the ratio between vapor mass and total mass.

- **Concentration** to define the amount of water vapor in the total volume. If this option is selected, a **Concentration** model input is automatically added to the **Model Inputs** section.
- **Moisture content** (default, also called mixing ratio or humidity ratio) to define the ratio between the mass of water vapor and the mass of dry air.
- **Relative humidity** ϕ_w , a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to air saturated with water vapor. The **Relative humidity, temperature condition** T_{ϕ_w} and **Relative humidity, absolute pressure condition** p_{ϕ_w} should be specified.

	Theory for Heat Transfer in Moist Air
	When the Porous Medium type is set to Local thermal nonequilibrium in the Porous Medium parent node, the Initial Values, Heat Source, Thermal Insulation, Symmetry (Heat Transfer Interface), Temperature, Heat Flux, Lumped System Connector, Phase Change Interface, Continuity, Inflow, Outflow, Open Boundary, Boundary Heat Source, Surface-to-Ambient Radiation (Heat Transfer Interface), and Deposited Beam Power features are available under the Fluid subnode. These subnodes allow the definition of domain and boundary conditions specific to the fluid phase temperature T_f.

DYNAMIC VISCOSITY

When the **Local thermal nonequilibrium** option is selected in the **Porous medium type** list of the **Porous Medium** parent node, and **Interstitial convective heat transfer coefficient** is set to **Spherical pellet bed**, the **Dynamic viscosity**, μ , should be set to evaluate the Nusselt number. Note that when **Fluid type** is set to **Moist air**, the moist air viscosity is defined by the node, and this section is not present.

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:

Geothermal Heating

Use this subnode under a [Porous Medium](#) node to take into account the contribution of geothermal heat production by adding a source term Q_{geo} to the right-hand side of the heat equation:

$$Q_{\text{geo}} = \rho_{\text{geo}} q_{\text{geo}} f(z_{\text{geo}})$$

The predefined expression of the heat source uses the **Geothermal Density**, the **Radiogenic heating per unit mass**, and a distribution function that can be set.

GEOTHERMAL HEAT PRODUCTION

Specify the **Radiogenic heating per unit mass** q_{geo} .

Select an option in the **Distribution type** list for the definition of $f(z_{\text{geo}})$:

- If **Uniform distribution** is selected, $f(z_{\text{geo}}) = 1$ and the geothermal heat source is assumed to be independent on depth.
- If **Exponential distribution** is selected, two additional input fields are displayed underneath. The exponential distribution $f(z_{\text{geo}}) = \exp(z_{\text{geo}}/h_{\text{geo}})$, is defined by the constant **Length scale**: h_{geo} and the variable **Depth** z_{geo} . The depth can be, for example, the vertical coordinate direction. The heat source achieves its maximum value where $z_{\text{geo}} = 0$, typically the top surface of a model.

GEOTHERMAL DENSITY

Select the **Geothermal density**: **Solids**, **Porous media**, or **User defined**:

- If **Solids** is selected, it calculates the geothermal density based on the volume fraction of solid material

$$\rho_{\text{geo}} = \frac{\sum_i \theta_{si} \rho_{si}}{\sum_i \theta_{si}}$$

- If **Porous media** is selected, it calculates the geothermal density based on all mobile and immobile components of the porous medium:

$$\rho_{\text{geo}} = \sum_i \theta_{si} \rho_{si} + \varepsilon_p \rho_f + \theta_{\text{imf}} \rho_{\text{imf}}$$

- 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

Select an option from the **Heat Source** list:

- For **General source** (default) enter a value for the distributed heat source Q_0 when the default option (**User defined**) is selected. See also [Additional General Source Options](#) to use predefined heat sources available from other interfaces.
- For **Linear source** enter a value for the **Production/absorption coefficient** q_s used in the predefined linear expression. The advantage of writing the source on this form is that it can be accounted for in the streamline diffusion stabilization. The stabilization applies when q_s is independent of the temperature, but some stability can be gained as long as q_s is only weakly dependent on the temperature.
- For **Heat rate** enter a value for the heat rate P_0 . In this case $Q_0 = P_0V$, 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 <i>COMSOL Multiphysics Reference Manual</i>
	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose Harmonic Perturbation. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.
	For the definition of a localized heat source, see Line Heat Source, Line Heat Source and Point Heat Source. For the definition of a heat on a boundary, see Boundary Heat Source.
	<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



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Heat Source

Heat Transfer in Porous Media > Porous Medium > Fluid > Heat Source

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Heat Source

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains > interface > Heat Source

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Heat Source

Heat Source (Pellets)

This node describes heat generation within the pellets (see [Heat Source](#) for more details). The domain selection inherits all valid domains from its parent feature. For the invalid domains, it shows not applicable.

Several **Heat Source** nodes can be used to account for different heat sources in different domains and pellets in the porous material.

PELLET

Select **All** or one of existing pellets for the **Heat Source** nodes.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Porous Media > Porous Medium > Pellets > Heat Source

Ribbon

Physics tab with **Pellets** selected in the model tree:

Attributes > Heat Source

Immobile Fluids (Porous Medium)

This node should be used to model an unsaturated porous matrix whose pore space is not solely filled with liquid and contains one or more gases considered to be immobile fluids; or a porous matrix containing insulated enclosures.



The **Immobile Fluids** node is available only when **Porous Medium type** is set to **Local thermal equilibrium** in the **Porous Medium** parent node, with either **Plane layers parallel to heat flow**, **Plane layers perpendicular to heat flow**, **Power law**, or **Equivalent thermal conductivity** option selected in the **Effective thermal conductivity** list.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

IMMOBILE FLUIDS

In this section, the total **Volume fraction** of the immobile fluids, θ_{imf} , should be set.

The total volume fraction of immobile fluids is calculated from

$$\theta_{\text{imf}} = \sum_i \theta_{\text{imfi}}$$

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

$$k_{\text{imf}} = \sum_i \theta_{\text{imf}i} k_{\text{imf}i}$$

In the **Porous Medium** node, the effective thermal conductivity is defined as:

- **Plane layers parallel to heat flow** (default), which calculates the effective conductivity of the solid-fluid system as the weighted arithmetic mean (or volume average) of the conductivities of the fluids and the porous matrix:

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

- **Plane layers perpendicular to heat flow**, which calculates the effective conductivity of the solid-fluid system as the weighted harmonic mean (or reciprocal average) of the conductivities of the fluids and the porous matrix:

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

- **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:

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



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 Porous Media > Porous Medium > Fluid > Initial Values

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Initial Values

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains > interface > Initial Values

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Initial Values

Initial Values (Pellets)

This node adds an initial value for the temperature of the pellets. The domain selection inherits all valid domains from its parent feature. For the invalid domains, it shows not applicable. In addition to the default **Initial Values** node always present in the interface, several **Initial Values** nodes can be used to set different initial values for different domains and pellets.

PELLET

This section is invisible for default **Initial Values** node. It is visible for later added **Initial Values** nodes. Select **All** or one of existing pellets for the nondefault **Initial Values** nodes.

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 Porous Media > Porous Medium > Pellets > Initial Values

Ribbon

Physics tab with **Pellets** selected in the model tree:

Attributes > Initial Values

Initial Values (Radiative Beam in Absorbing Medium Interface)

This node adds an initial value for the radiative intensity that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter a value or expression for the initial value of the **Radiative intensity I** (SI unit: W/m^2). This initial value is used by all the beams defined in the model.

LOCATION IN USER INTERFACE

Context Menus

Radiative Beam in Absorbing Media > Initial Values

Ribbon

Physics tab with **Radiative Beam in Absorbing Media** selected:

Domains > Initial Values

Initial Values (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node adds an initial value for the radiative intensity I (when **Discrete ordinates method** is selected as the **Radiation discretization method**) or the incident radiation G (when **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 over 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 over 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 6-15](#) in [Solid](#) node:

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

A user-defined value can be set for the initial irreversible transformation indicator in the [Initial Values](#) section.

User Defined

Enter values or expressions for the **Enthalpy change** L and the **Fraction of transformation** θ_{it} to define the heat source associated with the transformation as:

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

Specify Different Material Properties

Specific thermodynamics properties before and after complete transformation may be defined by selecting the **Specify different material properties for the transformed state** checkbox.

Choose a **Transformed material**, which can point to any material in the model. The default uses the **Domain material**. The properties before transformation are the ones specified in the **Heat Conduction, Solid** and **Thermodynamics, Solid** sections of the parent **Solid** node. The effective material properties are dynamically updated with the transformation evolution.

PARAMETERS

Overheating Analysis

Enter values for:

- **Transformation temperature** $T_{it, h}$ to define the (high) temperature that the solid needs to reach to start getting transformed.
- **Transformation time** $t_{it, h}$ to define the time needed for the complete transformation to happen while the temperature is above $T_{it, h}$.
- **Enthalpy change** $L_{it, h}$ to define the enthalpy variation associated with transformation due to overheating. The following heat source is added to the right-hand side of Equation 6-15 in **Solid** node:

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

Overcooling Analysis

Enter values for:

- **Transformation temperature** $T_{it, c}$ to define the (low) temperature that the solid needs to reach to start getting transformed.
- **Transformation time** $t_{it, c}$ to define the time needed for the complete transformation to happen while the temperature is below $T_{it, c}$.
- **Enthalpy change** $L_{it, c}$ to define the enthalpy variation associated with transformation due to overcooling. The following heat source is added to the right-hand side of Equation 6-15 in **Solid** node:

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

A user-defined value can be set for the initial irreversible transformation indicator in the [Initial Values](#) section.

HEAT CONDUCTION

This section is available when the **Specify different material properties for the transformed state** checkbox is selected.

Select a **Thermal conductivity** k_d — **From material** (default) or **User defined**, to be used for transformed solid. For **User defined** choose **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity and enter another value or expression in the field or matrix.

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics interface for example), you can select on which frame the **Thermal conductivity** k_d is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables $ht.k_{dxx}$, $ht.k_{dyy}$, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables $ht.k_{dxx}$, $ht.k_{dyy}$, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{d(x,y,z)} = \mathbf{R} k_{d(X,Y,Z)} \mathbf{R}^T$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

THERMODYNAMICS

This section is available when the **Specify different material properties for the transformed state** checkbox 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.

DISCRETIZATION

This section is available when the **Transformation model** is set to **Temperature threshold** or **Arrhenius kinetics**. To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog. You can change the solution algorithm by using the **Use local time integration** checkbox (cleared by default). When cleared, the solution of the equations for the auxiliary dependent variables in Time Dependent study will be governed by the time stepper algorithm used in the solver.



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 4-55](#).
- 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, nucleate boiling heat flux, and thermal contact. See Isothermal Domain Interface for more details. • Also see Theory for Lumped Isothermal Domain.
	<i>Natural Convection Cooling of a Vacuum Flask</i> : 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 > Isothermal Domain

Heat Transfer in Solids > Isothermal Domain

Heat Transfer in Fluids > Isothermal Domain

Heat Transfer in Porous Media > Isothermal Domain

Heat Transfer in Building Materials > Isothermal Domain

Bioheat Transfer > Isothermal Domain

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, or **Bioheat Transfer** selected:

Domains > interface > Isothermal Domain

Liquid Water (Moist Porous Medium)

This subnode defines the velocity field $\mathbf{u}_l$ and the saturation s_l of the liquid water phase used in Equation 6-6 of the [Moist Porous Medium](#) parent node. The density ρ_l (SI unit: kg/m^3), the heat capacity at constant pressure $C_{p,l}$ (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$), and the thermal conductivity k_l (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) are defined as functions of temperature.

LIQUID WATER PROPERTIES

The **Liquid water saturation**, s_l , used to evaluate the effective thermodynamics properties in the [Moist Porous Medium](#) parent node, should be set.

The default **Velocity field, liquid water, u_l** , is **User defined**. For **User defined**, enter values or expressions for the components based on space dimensions. Or select an existing velocity field in the component (for example, **Velocity field (spf)** from a **Laminar Flow** interface).

If the node was added automatically after selecting the predefined multiphysics interface **Heat and Moisture Transport**, the liquid saturation and liquid velocity of the multiphysics node **Heat and Moisture** are used by default, and the section is not editable. To edit the **Liquid water saturation** and **Velocity field, liquid water** fields, click **Make All Model Inputs Editable** ().

	• 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.

	Theory for Heat Transfer in Moist Air
	With some COMSOL products, the Viscous Dissipation (for heat generated by viscous friction), Pressure Work , and Convectively Enhanced Conductivity subnodes are available from the context menu (right-click the parent node) or from the Physics toolbar, Attributes menu.
	<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.

COORDINATE SYSTEM SELECTION

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate system**, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The subnodes of **Moist Porous Medium** inherit these coordinate system settings. In particular, the **Velocity field** (in [Moist Air \(Moist Porous Medium\)](#) subnode) and the

Dry bulk thermal conductivity and **Solid phase thermal conductivity** (in [Porous Matrix](#) ([Porous Medium](#), [Moist Porous Medium](#)) subnode) should be set according to the coordinate system selected in this section.

See [Coordinate Systems](#) in the *COMSOL Multiphysics Reference Manual* for more details.

POROUS MEDIUM MODEL SETTINGS

This section defines the averaging model for the computation of the **Effective thermal conductivity** k_{eff} , taking into account the properties of the solid matrix, moist air, and liquid water. The following options are available:

- **Plane layers parallel to heat flow** (default), which calculates the effective conductivity of the moist porous medium as the weighted arithmetic mean (or volume average) of the conductivities of the porous matrix, moist air, and liquid water:

$$k_{\text{eff}} = \theta_s k_s + \varepsilon_p [(1 - s_l) k_g + s_l k_l]$$

- **Plane layers perpendicular to heat flow**, which calculates the effective conductivity of the moist porous medium as the weighted harmonic mean (or reciprocal average) of the conductivities of the porous matrix, moist air, and liquid water:

$$\frac{1}{k_{\text{eff}}} = \frac{\theta_s}{k_s} + \frac{\varepsilon_p}{[(1 - s_l) k_g + s_l k_l]}$$

- **Power law**, which calculates the effective conductivity of the moist porous medium as the weighted geometric mean of the conductivities of the porous matrix, moist air, and liquid water:

$$k_{\text{eff}} = k_s^{\theta_s} \cdot [(1 - s_l) k_g + s_l k_l]^{\varepsilon_p}$$

- **Solid spherical inclusions**, which calculates the effective conductivity of the moist porous medium as:

$$k_{\text{eff}} = [(1 - s_l) k_g + s_l k_l] \frac{2[(1 - s_l) k_g + s_l k_l] + k_s - 2[(1 - s_l) k_g + s_l k_l] - k_s}{2[(1 - s_l) k_g + s_l k_l] + k_s + ([(1 - s_l) k_g + s_l k_l] - k_s)}$$

- **Fluid spherical inclusions**, which calculates the effective conductivity of the moist porous medium as:

$$k_{\text{eff}} = k_s \frac{2k_s + [(1 - s_l) k_g + s_l k_l] - 2(k_s - [(1 - s_l) k_g + s_l k_l]) \varepsilon_p}{2k_s + [(1 - s_l) k_g + s_l k_l] + (k_s - [(1 - s_l) k_g + s_l k_l]) \varepsilon_p}$$

- **Wrapped screen**, which calculates the effective conductivity of the moist porous medium as:

$$k_{\text{eff}} = [(1-s_l)k_g + s_l k_l] \frac{[(1-s_l)k_g + s_l k_l] + k_s - ([(1-s_l)k_g + s_l k_l] - k_s)\theta_s}{[(1-s_l)k_g + s_l k_l] + k_s + ([(1-s_l)k_g + s_l k_l] - k_s)\theta_s}$$

- **Sintered metal fibers**, which calculates the effective conductivity of the moist porous medium system as:

$$k_{\text{eff}} = \varepsilon_p^2 [(1-s_l)k_g + s_l k_l] + \theta_s^2 k_s + \frac{4\varepsilon_p \theta_s [(1-s_l)k_g + s_l k_l] k_s}{[(1-s_l)k_g + s_l k_l] + k_s}$$

If the porous matrix consists of several solids i of volume fraction θ_{si} and thermal conductivity k_{si} , the above equations are modified by replacing k_s by

$$\sum \theta_{si} k_{si} / \sum \theta_{si}, \text{ and } \theta_s \text{ by } \sum \theta_{si}.$$

It is also possible to directly define the equivalent thermal conductivity, k_{eq} . When **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list, a value for the **Equivalent thermal conductivity** k_{eq} should be specified directly. The default **Equivalent thermal conductivity** is taken **From material**. When a **Porous Material** node is active, the property of the **Homogenized Properties** section is used. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic**, enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

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 and Orbital Thermal Loads Interfaces)

This node enables to define the surface-to-surface radiation direction on boundaries surrounding the domains where it is defined. When several spectral bands are defined, the opacity can be set for all or only a selection of wavelengths.

When the **Emitted radiation direction** is defined by **Opacity controlled** in surface-to-surface boundary features ([Diffuse Mirror \(Surface-to-Surface Radiation Interface\)](#), [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#), [Prescribed Radiosity \(Surface-to-Surface Radiation Interface\)](#), [Opaque Surface \(Surface-to-Surface Radiation Interface\)](#), [Specular Mirror \(Surface-to-Surface Radiation Interface\)](#)), surface-to-surface radiation propagates in nonopaque domains. Alternatively the **Emitted radiation direction** can be defined using the normal orientation or on both sides of boundaries. In this case the **Opacity** node is ignored. Note that on boundaries where the [Semitransparent Surface \(Surface-to-Surface](#)

[Radiation Interface](#)) is applied, the **Emitted radiation direction** is **Both sides**, and the **Opacity** node is ignored.

By default, all the domains and the exterior of the Surface-to-Surface Radiation interface are considered as transparent. To set the exterior as opaque, change the **Selection** of the **Opacity** node to **All voids** in the **Domain Selection** section, and set it as **Opaque**. Depending on the geometry, an infinite void and several finite voids may be available for the selection. When the [Heat Transfer with Surface-to-Surface Radiation](#) multiphysics coupling is active, it controls the default opacity.

OPACITY

Depending on the **Wavelength dependence of radiative properties** defined in the [Radiation Settings](#) section of the physics interface settings, different sections display underneath.

When **Wavelength dependence of radiative properties** is set to **Constant**, select **Opaque** (default) 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**:

- When **Opacity** is set to **User defined for all bands** (default), set the opacity of the domain for all bands by selecting **Opaque** (default) or **Transparent**. Use this option if the transparency or opacity properties of the material are not wavelength dependent.
- When **Opacity** is set to **User defined for each band**, set the opacity for each spectral band by selecting the corresponding checkbox in the **Opaque** column of the table. By default the checkbox is selected for all spectral bands. Defining an opacity for each spectral band makes it possible to build models where the transparency or opacity properties of the material differs between spectral bands.

	<i>Free Convection in a Light Bulb</i> : Application Library path Heat_Transfer_Module/Thermal_Radiation/light_bulb
	<i>Thermo-Photo-Voltaic Cell</i> : 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.
	<i>Radiative Cooling of a Glass Plate</i> : 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 in the **Flux type** list are **General inward heat flux**, **Convective heat flux**, and **Nucleate boiling heat flux**. The settings are the same as for the [Heat Flux](#) node.

DOWNSIDE INWARD HEAT FLUX

The available options in the **Flux type** list are **General inward heat flux**, **Convective heat flux**, and **Nucleate boiling heat flux**. The settings are the same as for the [Heat Flux](#) node.

	The Out-of-Plane Heat Flux feature is disabled when at least one of the multiphysics couplings (Heat Transfer with Surface-to-Surface Radiation, Heat Transfer with Radiation in Participating Media, Heat Transfer with Radiative Beam in Absorbing Media) is active while solving on the same domain. This is noticed through a warning node displayed under Compile Equations. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.
	See Out-of-Plane Heat Transfer for the formulation of out-of-plane heat transfer in 1D, 1D axisymmetric, and 2D geometries. See also Out-of-Plane Domain Fluxes.
	Upside and downside settings can be visualized by plotting the global normal vector (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 <i>COMSOL Multiphysics Reference Manual</i>.
	<i>Out-of-Plane Heat Transfer for a Thin Plate</i>: Application Library path Heat_Transfer_Module/Verification_Examples/thin_plate

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Solids and Fluids > Out-of-Plane > Out-of-Plane Heat Flux**
- Heat Transfer in Porous Media > Porous Medium > Fluid > Out-of-Plane Heat Flux**
- Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Out-of-Plane Heat Flux**

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains > *interface* > **Out-of-Plane Heat Flux**

Physics tab with **Porous Medium** > **Fluid** or **Porous Medium** > **Porous Matrix** selected in the model tree:

Attributes > **Out-of-Plane Heat Flux**

Out-of-Plane Radiation

Out-of-plane heat transfer mechanism is used to reduce a model geometry to 2D or even 1D when the temperature variation is small in one or more directions; for example, when the object to model is thin or slender. This node models surface-to-ambient radiation on the upside and downside for the obtained 1D and 2D components. It adds the following contribution to the right-hand side of [Equation 6-8](#) or [Equation 6-9](#):

$$\varepsilon_u \sigma (T_{\text{amb, u}}^4 - T^4) + \varepsilon_d \sigma (T_{\text{amb, d}}^4 - T^4)$$

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 $T_{\text{amb}, d}$.

	The Out-of-Plane Radiation feature is disabled when at least one of the multiphysics couplings (Heat Transfer with Surface-to-Surface Radiation, Heat Transfer with Radiation in Participating Media, Heat Transfer with Radiative Beam in Absorbing Media) is active while solving on the same domain. This is noticed through a warning node displayed under Compile Equations. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.
	Out-of-Plane Heat Transfer
	Upside and downside settings can be visualized by plotting the global normal vector (nx, ny, nz), that always points from downside to upside. Note that the normal vector (ht.nx, ht.ny, ht.nz) may be oriented differently. See Tangent and Normal Variables in the <i>COMSOL Multiphysics Reference Manual</i>.
	<i>Out-of-Plane Heat Transfer for a Thin Plate</i>: Application Library path Heat_Transfer_Module/Verification_Examples/thin_plate

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Solids and Fluids > Out-of-Plane > Out-of-Plane Radiation**
- Heat Transfer in Porous Media > Porous Medium > Fluid > Out-of-Plane Radiation**
- Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Out-of-Plane Radiation**

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Domains > *interface* > Out-of-Plane Radiation

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Out-of-Plane Radiation

Participating Medium (Radiation in Participating Media Interface)

This node should be used when radiation occurs in a medium not completely transparent, in which the radiation rays interact with the medium. It calculates the heating due to the propagation of the rays, and takes into account the absorbed, the emitted, and the scattered radiation, depending on the **Radiation discretization method** selected in the [Participating Media Settings](#) section of the interface.

It calculates the radiative heat source term Q (SI unit: W/m^3), defined by:

$$Q = \kappa(G - 4\pi I_b(T))$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2).
- $I_b(T)$ is the blackbody radiative intensity (SI unit: $\text{W}/(\text{m}^2 \cdot \text{sr})$), 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: $\text{W}/(\text{m}^2 \cdot \text{K}^4)$).
- 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: $\text{W}/(\text{m}^2 \cdot \text{sr})$) 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

$$G = \sum_{i=1}^{N_V} \omega_i I_i \quad (6-10)$$

and, in a cartesian coordinates system

$$\mathbf{S}_i \cdot \nabla I_i = \kappa I_b(T) - \beta I_i + \frac{\sigma_s}{4\pi} \sum_{j=1}^{N_V} \omega_j I_j \phi(\mathbf{S}_j, \mathbf{S}_i) \quad (6-11)$$

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 j^{th} quadrature weight.

If the **Radiation discretization method** is **P1 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
Scattering modeling	Linear Polynomial (in 2D and 3D)	Linear
Computational cost	High: up to 80 additional degrees of freedom (I_i)	Medium: 1 additional degree of freedom (G)

If radiative emission may be neglected, the [Absorbing-Scattering Medium \(Radiation in Absorbing-Scattering Media Interface\)](#) node may be used instead. The table below describes the different effects accounted for by the interfaces found under the **Heat Transfer > Radiation** branch (☀️).

TABLE 6-4: RADIATION EFFECTS CALCULATED BY THE RADIATION INTERFACES.

RADIATION EFFECT	RPM	RASM	RBAM
Absorption	Yes	Yes	Yes
Scattering	Yes	Yes	No
Emission	Yes	No	No

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when temperature-dependent material properties are used. The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics

interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), 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 `rpm.T`.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength may be accessed through the `rpm.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 rpm.T .

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength may be accessed through the rpm.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:

$$K = \frac{1}{4\pi} \cdot \int \frac{1 - \eta^2}{(1 + \eta^2 - 2\eta\mu_0)^{3/2}} d\Omega$$

For **Linear anisotropic** and **Polynomial anisotropic**, select the **Normalize phase function** checkbox to define a phase function such as

$$\int\limits_{4\pi} \phi(\mu_0)d\Omega = 4\pi$$

The normalization is automatically applied for the **Henyeey-Greenstein** option.

For **Linear anisotropic**, **Polynomial anisotropic**, and **Henyeey-Greenstein**, select the **Wavelength-dependent scattering type** checkbox to average each **Legendre coefficient** $a_1, \dots, a_{12}$ or the **Anisotropy parameter** μ to obtain piecewise constant coefficients on each spectral band.

	• Theory for Radiation in Participating Media • Discrete Ordinates Method (DOM) • P1 Approximation Theory
	<i>Radiative Heat Transfer in a Utility Boiler:</i> Application Library path Heat_Transfer_Module/Thermal_Radiation/boiler <i>Radiative Cooling of a Glass Plate:</i> Application Library path Heat_Transfer_Module/Thermal_Radiation/glass_plate <i>Radiative Heat Transfer in Finite Cylindrical Media:</i> Application Library path Heat_Transfer_Module/Verification_Examples/cylinder_participating_media <i>Radiative Heat Transfer in Finite Cylindrical Media — P1 Method:</i> Application Library path Heat_Transfer_Module/Verification_Examples/cylinder_participating_media_p1

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

Pellets (Porous Medium)

This node defines the porosity and material properties of the pellets contained in a packed bed, used in the heat transfer equations of the [Porous Medium](#) parent node, when the **Packed bed** option is selected in the **Porous medium type** list. The coupling condition between the pellets and the fluid phase of the packed bed can be specified in the [Pellet-Fluid Interface \(Porous Medium\)](#) subnode.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (), 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.

POROSITIES

The default **Bed porosity** ε_p of the packed bed is taken **From material**. In a **Porous Material** node, the porosity is defined as follows:

$$\varepsilon_p = 1 - \sum_i \theta_{si} - \sum_i \theta_{pe} - \sum_i \theta_{imfi}$$

where θ_{si} , θ_{pe} , and θ_{imfi} are the volume fractions of the **Solid**, **Pellet**, and **Immobile Fluid** subnodes under the **Porous Material** node.

For **User defined**, enter a value or expression.

As porosity can be considered within the pellets, set the **Pellet porosity**, ε_{pe} . By default it is taken **From material**. Go to the **Pellet** subnode of the **Porous Material** node to enter a value or expression.

HEAT CONDUCTION, PELLETS

Set the thermal conductivity of the porous matrix within the pellets, k_{pe} . It is used to compute the effective thermal conductivity of the pellet filled with fluid, $k_{pe,eff}$, defined by

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

The default **Thermal conductivity** k_{pe} is taken **From material**. For **User defined**, enter a value or expression. A scalar value should be set as it is used in the 1D microscale equation.

THERMODYNAMICS, PELLETS

This section defines the thermodynamics properties of the porous matrix within the pellets.

The specific heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit of mass of the pellets.

The **Density** ρ_{pe} and the **Specific heat capacity** $C_{p,pe}$ should be specified. For **From Material** option, see [Material Density in Features Defined in the Material Frame](#) if a temperature-dependent density should be set.

These properties are used to compute the effective volumetric heat capacity at constant pressure inside the pellet filled with fluid, $(\rho C_p)_{pe,eff}$, defined by

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

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Porous Media > Porous Medium > Pellets

Ribbon

Physics tab with **Porous Medium** selected in the model tree:

Attributes > Pellets

Pellet-Fluid Interface (Porous Medium)

This node defines the coupling condition between the fluid phase and the pellets, when the **Packed bed** option is selected in the **Porous medium type** list of the [Porous Medium](#) parent node.

PELLET

This section is invisible for the default **Pellet-Fluid Interface** node.

Select All or one of existing pellets for the nondefault **Pellet-Fluid Interface** nodes. To define properties of a pellet phase in the porous medium use a **Pellet** subnode of a **Porous Material** node (see [Pellet](#) in the *COMSOL Multiphysics Reference Manual*).

PELLET-FLUID INTERFACE

Select an option in the **Coupling condition** list to couple the macroscale heat equation solved in the fluid phase and the microscale equation solved in the pellets to account for radial thermal conduction. The **Coupling condition** list is selectable for the default **Pellet-Fluid Interface** node, but it is disabled for nondefault **Pellet-Fluid Interface** nodes because it is supposed that there should be only one type of boundary condition in a packed bed.

With the **Continuous temperature** option (default), the temperature at the outer surface of the pellets and the fluid temperature are enforced to be equal, and the resulting heat flux is applied in the fluid and pellets equations. See the [Constraint Settings](#) section below for advanced settings relative to the continuity constraint.

With the **Convective heat flux** option, a heat transfer term, proportional to the temperature difference between the fluid phase and the outer surface of the pellets, is applied in the equations of both phases. The heat transfer term is applied in an averaged way in the fluid equation, by multiplying it by the **Specific surface area**, S_b . Either use the **Automatic** expression for S_b , defined as a function of the average pellet diameter d_{pe} , or set a **User defined** value or expression. Either use the **Automatic** option

for the **Interstitial heat transfer coefficient**, $h_{pe,f}$, defined as a function of the average pellets diameter d_{pe} and the fluid-to-solid Nusselt number for which the fluid dynamic viscosity μ is needed (and should be set in the [Fluid \(Porous Medium\)](#) subnode). Or set a **User defined** value or expression for $h_{pe,f}$.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog. The settings of this section have no effect when the **Coupling condition** is **Convective heat flux** in the [Pellet-Fluid Interface](#) section.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Porous Media > Porous Medium > Pellet-Fluid Interface

Ribbon

Physics tab with **Porous Medium** selected in the model tree:

Attributes > Pellet-Fluid Interface

Phase Change Material

This subnode is used to specify the properties of a phase change material when modeling heat transfer in solids, fluids, and porous media. It uses the apparent heat capacity method which describes the phase transition within a temperature interval where latent heat is included as an additional term in the heat capacity – hence, the name of the method. 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. 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 a **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

In this section, the characteristics of the phase transitions from phase j to phase $j + 1$ are defined. This means that the phase transition function $\alpha_{j \rightarrow j+1}$ is specified, which at the same time corresponds to the volume fraction of phase $j + 1$.

The **Phase transition function** describes how the transition from phase j to phase $j+1$ occurs. **Heaviside** (the default) describes a transition as a smooth step function with continuous derivatives up to second order and **Linear** a transition as a linear function. Each transition is assumed to occur in a temperature interval $\Delta T_{j \rightarrow j+1}$ around the phase change temperature $T_{pc, j \rightarrow j+1}$. During phase change between the temperatures $T_{pc, j \rightarrow j+1} - \Delta T_{j \rightarrow j+1}/2$ and $T_{pc, j \rightarrow j+1} + \Delta T_{j \rightarrow j+1}/2$, a total heat per unit volume equal to $L_{j \rightarrow j+1}$ is released.

With **User defined**, an arbitrary function for $\alpha_{j \rightarrow j+1}$ can be defined. The default expression is `f1c2hs(T-273.15[K],5[K])`, which corresponds to the default Heaviside phase transition function.

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.



The transition interval between two phases has to be expressed in the temperature unit that corresponds to the unit system chosen for the model. By default, the unit system is SI which indicates that the interval has to be given in K. Using a value expressed in degrees Celsius is likely to produce unexpected behavior. For example, a transition interval set to 1 [degC] would be interpreted as 274.15 K.

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 and at a temperature of 273.15 K.

When choosing a user define phase transition function, define the **Phase transition between phase 1 and phase 2** directly. Enter an expression for the phase transition function $\alpha_{1 \rightarrow 2}$ that explicitly describes the volume fraction of phase 2. Enter any additional functions as per the **Number of phase transitions**. The values of $\alpha_{1 \rightarrow 2}$ should lie in the interval from 0 to 1. The default expression is `f1c2hs(T-273.15[K],5[K])` which corresponds to the default Heaviside phase transition function. See the [Phase Transition Function](#) section in the [Theory for Heat Transfer with Phase Change](#) chapter for more details about the definition of $\alpha_{1 \rightarrow 2}$.

About the Phases

When using Heaviside or linear for the phase transition function, the phases are ordered according to the temperatures of fusion. Hence, the material properties of phase 1 are used when $T < T_{pc, 1 \rightarrow 2}$, while the material properties of phase 2 hold for $T > T_{pc, 1 \rightarrow 2}$. Within the phase transition interval the properties are mixed according to the phase transition function.

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 or a **Porous Matrix** subnode of a **Porous Medium** feature, the following material properties should be set:

- **Thermal conductivity** k_i . The default uses the material values for phase i . For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics interface for example), you can select on which frame the **Thermal conductivity** k_i is specified, with the **Deformation model for thermal conductivity** option, see [Heat Conduction, Solid](#) for details.
- **Heat capacity at constant pressure** $C_{p, i}$. The default is 4200 J/(kg·K).

Note that these settings override the settings of the **Heat Conduction, Solid** and **Thermodynamics, Solid** sections of the parent **Solid** node.

When the **Phase Change Material** subnode is added under a **Fluid** node or a **Fluid** subnode of a **Porous Medium** feature, 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 **Automatic**, it calculates the ratio of specific heats by using Mayer’s relation:

$$\gamma = \frac{\rho C_p \chi_t}{\rho C_p \chi_t - T \alpha_p^2}$$

considering that:

- α_p is the coefficient of thermal expansion (SI unit: 1/K):

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

- χ_t is the isothermal compressibility (SI unit: 1/Pa):

$$\chi_t = -\frac{1}{\rho} \frac{\partial \rho}{\partial p}$$

Note that these settings override the settings of the **Heat Conduction, Fluid** and **Thermodynamics, Fluid** sections of the parent node.

	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 <i>COMSOL Multiphysics Reference Manual</i>) can be used to take into account model deformation.
	<i>Phase Change</i>: Application Library path Heat_Transfer_Module/Phase_Change/phase_change <i>Continuous Casting — Apparent Heat Capacity Method</i>: Application Library path Heat_Transfer_Module/Thermal_Processing/continuous_casting_apparent_heat_capacity <i>Cooling and Solidification of Metal</i>: Application Library path Heat_Transfer_Module/Thermal_Processing/cooling_solidification_metal

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Solids > Solid > Phase Change Material**
- Heat Transfer in Fluids > Fluid > Phase Change Material**
- Heat Transfer in Porous Media > Porous Medium > Phase Change Material**
- Heat Transfer in Porous Media > Porous Medium > Fluid > Phase Change Material**
- Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Phase Change Material**

Ribbon

Physics tab with **Solid**, **Fluid**, **Porous Medium > Fluid**, or **Porous Medium > Porous Matrix**, selected in the model tree:

Attributes > Phase Change Material

Porous Matrix (Porous Medium, Moist Porous Medium)

This node defines the porosity and material properties of the solid matrix used in the heat transfer equations of the [Moist Porous Medium](#) and [Porous Medium](#) parent

nodes, to model heat transfer in a porous matrix, possibly consisting of several solids, and filled with a mobile fluid, and one or more immobile fluids.



- The velocity field and material properties of the mobile fluid can be specified in the [Fluid \(Porous Medium\)](#) subnode, by defining it as a general gas or liquid, as an ideal gas, or as moist air.
- The total volume fraction and material properties of immobile fluids can be specified in the [Immobile Fluids \(Porous Medium\)](#) subnode.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

Other options are **User defined** and all the temperature variables from the physics interfaces included in the model.



This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and

in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

MATRIX PROPERTIES

The default **Porosity** ε_p of the solid matrix is taken **From material**. In a **Porous Material** node, the porosity is defined as follows:

$$\varepsilon_p = 1 - \sum_i \theta_{si} - \sum_i \theta_{imfi}$$

where θ_{si} and θ_{imfi} are the volume fractions of the **Solid** and **Immobile Fluids** subnodes under the **Porous Material** node.

For **User defined**, enter a value or expression.

Choose to define either the **Dry bulk properties** or the **Solid phase properties** for thermal conductivity, density, and heat capacity at constant pressure of the porous matrix. The first option sets the properties of the matrix including empty pores, while the second option defines the properties of the pure solid phase.

HEAT CONDUCTION, POROUS MATRIX

The thermal conductivity k_s describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k_s \nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k_s is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full**, based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic**, enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

When the thermal conductivity is taken **From material** and more than one **Solid** subnode is added under the **Porous Material** node, the average property is obtained by applying a volume average model:

$$k_s = \sum_i \theta_{si} k_{si}$$

Alternatively, set the **Dry bulk thermal conductivity** k_b , when **Define** is set to **Dry bulk properties** in the **Matrix Properties** section. The corresponding property for the solid phase is then defined as

$$k_s = \frac{k_b}{\theta_s}$$

Note that this section is not available when **Porous medium type** is set to **Local thermal equilibrium** and **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list of the **Porous medium** parent node.

THERMODYNAMICS, POROUS MATRIX

This section defines the thermodynamics properties of the porous matrix.

The specific heat capacity describes the amount of heat energy required to produce a unit temperature change in a unit of mass of the solid material.

The **Density** ρ_s and the **Specific heat capacity** $C_{p,s}$ should be specified. For **From Material** option, see [Material Density in Features Defined in the Material Frame](#) if a temperature-dependent density should be set.

When the density and specific heat capacity are taken **From material** and more than one **Solid** subnode is added under the **Porous Material** node, the effective properties are obtained by applying a volume averaging model:

$$\rho_s = \sum_i \theta_{si} \rho_{si}$$

$$C_{p,s} = \frac{\sum_i \theta_{si} \rho_{si} C_{p,si}}{\sum_i \theta_{si} \rho_{si}}$$

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

$$C_{p,s} = \frac{C_{p,b}}{\theta_s}$$

	When the porosity ε_p is taken From material, the solid volume fraction θ_s is defined in the material, under the Solid subnode. In case of multiple solids, it is needed to refer to the material, as it is not possible to deduce the values of θ_{si} for the different solids from the porosity. When the porosity is User defined, the volume fraction of each solid θ_{si} is obtained by dividing $\theta_s = 1 - \varepsilon_p$ by the number of solids.
	When the Porous Medium type is set to Local thermal nonequilibrium in the Porous Medium parent node, the Initial Values, Heat Source, Thermal Insulation, Symmetry (Heat Transfer Interface), Temperature, Heat Flux, Lumped System Connector, Continuity, Boundary Heat Source, Surface-to-Ambient Radiation (Heat Transfer Interface), and Deposited Beam Power features are available under the Porous Matrix subnode. These subnodes allow the definition of domain and boundary conditions specific to the fluid phase temperature T_s.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Porous Media > Porous Medium > Porous Matrix

Heat Transfer in Moist Porous Media > Moist Porous Medium > Porous Matrix

Ribbon

Physics tab with **Porous Medium** or **Moist Porous Medium** selected in the model tree:

Attributes > Porous Matrix

Porous Medium

This node models heat transfer in a porous matrix, possibly consisting of several solids, and filled with a mobile fluid, and one or more immobile fluids. It allows to make the local thermal equilibrium assumption or not, and to handle the case of a packed bed of pellets in a specific way, through the **Porous medium type** option.

Local thermal equilibrium

With this assumption, one single equation is solved for both phases, using effective material properties. See [Equation 4-41](#) for details.

In particular, when no immobile fluid is present in the pore space, the effective volumetric heat capacity at constant pressure is defined as

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

If the porous matrix consists of several solids i of volume fraction θ_{si} , heat capacity $C_{p,si}$, and density ρ_{si} , the above equation is modified as follows:

$$(\rho C_p)_{\text{eff}} = \sum_i \theta_{si} \rho_{si} C_{p,si} + \varepsilon_p \rho_f C_{p,f}$$

See [Immobile Fluids \(Porous Medium\)](#) to take into account the presence of immobile fluids in the expression of effective volumetric heat capacity at constant pressure.

See [Porous Medium](#) for the definition of the effective thermal conductivity.

Local Thermal Nonequilibrium

When the solid and fluid temperatures are not in equilibrium, two heat equations are solved in the solid and fluid subdomains, and coupled through a transfer term proportional to the temperature difference between both phases. See [Equation 4-43](#) and [Equation 4-44](#) for details about the heat transfer equation in each phase, and [Porous Medium](#) for the definition of the transfer term between fluid and solid phases.

Packed Bed

When the porous medium is a packed bed of pellets, and when the thermal conductivity of the pellets is small enough compared to the fluid conductivity to neglect heat transfer between the pellets, the macroscale heat equation for the solid phase of the standard local thermal nonequilibrium model is replaced by a microscale equation for the radial variation of the temperature due to conduction in the pellets.

See [Equation 4-45](#) and [Equation 4-46](#) for details about the heat transfer equation in each phase, and [Pellet-Fluid Interface \(Porous Medium\)](#) for the definition of the coupling condition between the fluid phase and pellets equations.

COORDINATE SYSTEM SELECTION

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate**

system, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The subnodes of **Porous Medium** inherit these coordinate system settings. In particular, the **Velocity field** and **Thermal Conductivity** (in [Fluid \(Porous Medium\)](#) subnode) and the **Dry bulk thermal conductivity** and **Solid phase thermal conductivity** (in [Porous Matrix \(Porous Medium, Moist Porous Medium\)](#) subnode) should be set according to the coordinate system selected in this section.

See [Coordinate Systems](#) in the *COMSOL Multiphysics Reference Manual* for more details.

POROUS MEDIUM

Select between the **Local thermal equilibrium**, the **Local thermal nonequilibrium**, and the **Packed bed** options in the **Porous medium type** list.

Depending upon the selected option, further settings are required underneath: either the effective thermal conductivity required by the single heat transfer equation, or the parameters used to couple the two heat transfer equations in each phase. See the [Local thermal equilibrium](#) and [Local Thermal Nonequilibrium](#) sections below for details.

It also changes the availability of some subnodes:

- The **Geothermal Heating**, **Immobile Fluids**, and **Optically Thick Participating Medium** subnodes are available under **Porous Medium** only when the **Porous Medium type** is set to **Local thermal equilibrium**.
- The [Initial Values](#), [Heat Source](#), [Thermal Insulation](#), [Symmetry \(Heat Transfer Interface\)](#), [Temperature](#), [Heat Flux](#), [Lumped System Connector](#), [Phase Change Interface](#), [Continuity](#), [Inflow](#), [Outflow](#), [Open Boundary](#), [Boundary Heat Source](#), [Surface-to-Ambient Radiation \(Heat Transfer Interface\)](#), and [Deposited Beam Power](#) features are available under the [Fluid \(Porous Medium\)](#) subnode only when the **Porous Medium type** is set to **Local thermal nonequilibrium** or **Packed bed**. These subnodes allow the definition of domain and boundary conditions specific to the fluid phase temperature T_f .
- The [Initial Values](#), [Heat Source](#), [Thermal Insulation](#), [Symmetry \(Heat Transfer Interface\)](#), [Temperature](#), [Heat Flux](#), [Lumped System Connector](#), [Continuity](#), [Boundary Heat Source](#), [Surface-to-Ambient Radiation \(Heat Transfer Interface\)](#), and [Deposited Beam Power](#) features are available under the [Porous Matrix \(Porous Medium, Moist Porous Medium\)](#) subnode only when the **Porous Medium type** is set to **Local thermal nonequilibrium**. These subnodes allow the definition of domain and boundary conditions specific to the solid phase temperature T_s .

Local Thermal Equilibrium

When **Local thermal equilibrium** is selected in the **Porous medium type** list, this section defines the **Effective thermal conductivity** taking into account the properties of the solid matrix and the mobile fluid. The following averaging models are available:

- **Plane layers parallel to heat flow** (default), which calculates the effective conductivity of the solid-fluid system as the weighted arithmetic mean (or volume average) of the conductivities of the fluid and the porous matrix:

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

- **Plane layers perpendicular to heat flow**, which calculates the effective conductivity of the solid-fluid system as the weighted harmonic mean (or reciprocal average) of the conductivities of the fluid and the porous matrix:

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

- **Power law**, which calculates the effective conductivity of the solid-fluid system as the weighted geometric mean of the conductivities of the fluid and the porous matrix:

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

- **Solid spherical inclusions**, which calculates the effective conductivity of the solid-fluid system as:

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

- **Fluid spherical inclusions**, which calculates the effective conductivity of the solid-fluid system as:

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

- **Wrapped screen**, which calculates the effective conductivity of the solid-fluid system as:

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

- **Sintered metal fibers**, which calculates the effective conductivity of the solid-fluid system as:

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

If the porous matrix consists of several solids i of volume fraction θ_{si} and thermal conductivity k_{si} , the above equations are modified by replacing k_s by $\sum_i \theta_{si} k_{si} / \sum_i \theta_{si}$, and θ_s by $\sum_i \theta_{si}$.



The **Immobile Fluids** subnode is available only when **Plane layers parallel to heat flow**, **Plane layers perpendicular to heat flow**, **Power law**, or **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list.

It is also possible to define directly the equivalent thermal conductivity, k_{eq} . When **Equivalent thermal conductivity** is selected in the **Effective thermal conductivity** list, a value for the **Equivalent thermal conductivity** k_{eq} should be specified directly. The default **Equivalent thermal conductivity** is taken **From material**. When a **Porous Material** node is active, the property of the **Homogenized Properties** section is used. For **User defined**, select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic**, enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

Local Thermal Nonequilibrium

When **Local thermal nonequilibrium** is selected in the **Porous medium type** list, select an **Interstitial convective heat transfer coefficient: Spherical pellets**, **General configuration**, or **User defined** (the default).

Spherical Pellets In this particular configuration, the interstitial convective heat transfer coefficient can be directly expressed as a function of the average pellets diameter d_{pe} and the fluid-to-solid Nusselt number for which the fluid dynamic viscosity μ is needed.

Enter a value for the **Average diameter** d_{pe} (SI unit: m) of the pellets. Default value is 10^{-3} m.

The dynamic viscosity needed to evaluate the Nusselt number is set in the [Fluid \(Porous Medium\)](#) subnode. See [Dynamic Viscosity](#) for details.

Note that with this option, the radial variation of the temperature within the pellets is not precisely accounted for. Change to the **Packed bed** option (in the **Porous medium**

type list) to solve for a specific equation for radial conduction in the pellets. See [Packed Bed](#) for details.

General Configuration The interstitial convective heat transfer coefficient is expressed as the product of the specific surface area S_b and the interstitial heat transfer coefficient h_{sf} .

Enter a value for the **Specific surface area** S_b (SI unit: $1/m$).

Enter a value for the **Interstitial heat transfer coefficient** h_{sf} (SI unit: $W/(m^2 \cdot K)$).

User Defined Enter a custom value for q_{sf} (SI unit: $W/(m^3 \cdot K)$).



- See [Immobile Fluids \(Porous Medium\)](#) to take into account the presence of immobile fluids in the expression of effective thermal conductivity in the local thermal equilibrium case.
- The velocity field and material properties of the mobile fluid can be specified in the [Fluid \(Porous Medium\)](#) subnode, by defining it as a general gas or liquid, as an ideal gas, or as moist air.
- The porosity ϵ_p and material properties of the solid matrix can be specified in the [Porous Matrix \(Porous Medium, Moist Porous Medium\)](#) subnode.
- The packed bed and pellets porosities ϵ_p and ϵ_{pe} , and the material properties of the pellets can be specified in the [Pellets \(Porous Medium\)](#) subnode.
- The coupling condition between the fluid phase and the pellets can be specified in the [Pellet-Fluid Interface \(Porous Medium\)](#) subnode.
- When the porosity ϵ_p is taken **From material**, the solid volume fraction θ_s is defined in the material, under the **Solid** subnode. In case of multiple solids, it is needed to refer to the material, as it is not possible to deduce the values of θ_{si} for the different solids from the porosity. When the porosity is **User defined**, the volume fraction of each solid θ_{si} is obtained by dividing $\theta_s = 1 - \epsilon_p$ by the number of solids.



See [Porous Material](#) in the *COMSOL Multiphysics Reference Manual*.

	Theory for Heat Transfer in Porous Media
	With some COMSOL products, the Thermal Dispersion , Viscous Dissipation , and Geothermal Heating subnodes are available from the context menu (right-click the parent node) or from the Physics toolbar, Attributes menu.
	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Porous Medium

Heat Transfer in Solids > Specific Media > Porous Medium

Heat Transfer in Fluids > Specific Media > Porous Medium

Heat Transfer in Porous Media > Specific Media > Porous Medium

Heat Transfer in Building Materials > Specific Media > Porous Medium

Ribbon

Physics tab with **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, or **Heat Transfer in Building Materials** selected:

Domains > Specific Media > Porous Medium

Pressure Work

This subnode adds the following contribution to the right-hand side of the heat transfer equation to model the result of heating under adiabatic compression as well as some thermoacoustic effects:

$$Q_p = \alpha_p T \left(\frac{\partial p_A}{\partial t} + \mathbf{u} \cdot \nabla p_A \right) \quad (6-13)$$

where α_p is the coefficient of thermal expansion defined as:

$$\alpha_p = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right)_p$$

The pressure work is calculated using the relative pressure, and is generally small for low Mach number flows.

No settings are required.

	When the Nonisothermal Flow multiphysics coupling node is added, the effect of pressure work is automatically taken into account. In this case, the Pressure Work feature is overridden by the multiphysics coupling node's contribution.
	Theory for Heat Transfer in Fluids
	A similar term can be included to take into account thermoelastic effects in solids. See Thermoelastic Damping .

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Fluids > Fluid > Pressure Work**
- Heat Transfer in Moist Air > Moist Air > Pressure Work**
- Heat Transfer in Porous Media > Porous Medium > Fluid > Pressure Work**

Ribbon

Physics tab with **Fluid** or **Porous Medium > Fluid** selected in the model tree:

Attributes > Pressure Work

Radiative Source

This node models a radiative heat source within the domain.

When the **Radiation discretization method** is **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 [Absorbing Medium \(Radiative Beam in Absorbing Medium Interface\)](#) to the spatial frame. See [Conversion Between Material and Spatial Frames](#) for details.
- The **Nonsolid** option defines Q_0 in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.



This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See [Handling Frames in Heat Transfer](#) and [Material and Spatial Frames](#) for details.

RADIATIVE SOURCE

Select a **Radiative source type** from the list:

- For **Power density** enter a value for the power density Q_0 . When the **Radiation discretization method** is **Discrete ordinates method**, Q_0 is divided by 4π to obtain the directional power density $q_{0,j}$ to be added in the radiative heat transfer equation. When the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the power density can also be **User defined for each band**, or set as a **User defined distribution**.

- For **Directional power density** (available when the **Radiation discretization method** is **Discrete ordinates method**) enter directly a value for the directional power density $q_{0,j}$. When the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the directional power density can also be **User defined for each band**, or set as a **User defined distribution**.
- For **Energy rate** enter a value for the energy rate P_0 . In this case $Q_0 = P_0 V$, where V is the total volume of the selected domains. When the **Radiation discretization method** is **Discrete ordinates method**, Q_0 is divided by 4π to obtain the directional power density $q_{0,j}$ to be added in the radiative heat transfer equation. When the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the energy rate can also be **User defined for each band**, or set as a **User defined distribution**.

When **User defined for each band** is selected in any of the options above, set a value for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength.

When **User defined distribution** is selected in any of the options above, set a value or expression. Any expression is then averaged on each spectral band to obtain a piecewise constant function. If the average value on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

LOCATION IN USER INTERFACE

Context Menus

Radiation in Participating Media > Radiative Source

Radiation in Absorbing-Scattering Media > Radiative Source

Ribbon

Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Domains > Radiative Source

Shape Memory Alloy

This node models heat transfer in shape memory alloys, and takes into account the martensite and austenite composition. This feature is designed to be coupled with the **Shape Memory Alloy** feature of the Structural Mechanics interface which calculates the alloy composition from the mechanical and thermal loads. Changes in martensite and austenite composition modify the alloys thermal properties (thermal conductivity and

heat capacity), and release (from austenite to martensite state) or absorb (from martensite to austenite state) energy.

It uses this version of the heat equation to model heat transfer in alloys:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (6-14)$$

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

with the following material properties, fields, and sources:

- ρ (SI unit: kg/m^3) is the alloy's density.
- C_p (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) 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 **Moving Mesh** node 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:

$$k = k_M^\xi \cdot k_A^{(1 - \xi)}$$

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics interface for example), you can select on which frame the thermal conductivities of the austenite and martensite phases are specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivities are supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the effective conductivity variables. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivities are supposed to be given on the spatial frame. In case of isotropic materials, the effective thermal conductivity variables are directly equal to the weighted average of the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{(x,y,z)} = \mathbf{R} k_{(X,Y,Z)} \mathbf{R}^T$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

AUSTENITE

Select any component material from the list to define the austenite material properties. The default uses the **Domain material**.

The default **Thermal conductivity** k_A is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic** enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

The default **Heat capacity at constant pressure** $C_{p,A}$ 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.



When the material properties for the austenite and martensite phases are defined using the **From material** option, the properties are taken from **Austenite phase** and **Martensite phase** property groups respectively. If a property is not defined in these groups but is available in the **Basic** group, its definition is automatically synchronized. This mechanism is designed to make it possible to use two materials, one pure austenite material and one pure martensite material with properties defined in the basic group as an alternative to a single material with the properties defined in the **Austenite phase** and **Martensite phase** property groups. In that case, the two materials have to be selected instead of using the default **Domain material** option.

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:

$$\rho C_p \frac{\partial T}{\partial t} + \rho C_p \mathbf{u} \cdot \nabla T + \nabla \cdot \mathbf{q} = Q \quad (6-15)$$

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

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 **Moving Mesh** node when parts of the model are moving in the material frame.
- Q (SI unit: W/m^3) is the heat source (or sink). Add one or more heat sources as separate physics features. See [Heat Source](#) node and [Thermoelastic Damping](#) subnode for example.

For a steady-state problem the temperature does not change with time and the first term disappears.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.



This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

HEAT CONDUCTION, SOLID

The thermal conductivity k describes the relationship between the heat flux vector $\mathbf{q}$ and the temperature gradient ∇T in $\mathbf{q} = -k\nabla T$, which is Fourier's law of heat conduction. Enter this quantity as power per length and temperature.

The default **Thermal conductivity** k is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. For **Isotropic** enter a scalar which will be used to define a diagonal tensor. For the other options, enter values or expressions into the editable fields of the tensor.

The components of the thermal conductivity k when given on tensor form (k_{xx} , k_{yy} , and so on, representing an anisotropic thermal conductivity) are available as `ht.kxx`, `ht.kyy`, and so on (using the default name `ht`). The single scalar mean effective thermal conductivity `ht.kmean` is the mean value of the diagonal elements k_{xx} , k_{yy} , and k_{zz} .

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics interface for example), you can select on which frame the thermal conductivity is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables `ht.kxx`, `ht.kyy`, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables `ht.kxx`, `ht.kyy`, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{(x,y,z)} = \mathbf{R}k_{(X,Y,Z)}\mathbf{R}^T$$

where **R** is the rotation matrix between the material and the spatial frames.



Fourier’s law assumes that the thermal conductivity tensor is symmetric. A nonsymmetric tensor can lead to unphysical results.

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.alphaTdxx`, `ht.alphaTdy`, and so on (using the default physics name `ht`). The single scalar mean thermal diffusivity `ht.alphaTdMean` is the mean value of the diagonal elements α_{xx} , α_{yy} , and α_{zz} . The denominator ρC_p is the effective volumetric heat capacity which is also available as a predefined quantity, `ht.C_eff`.



- [Theory for Heat Transfer in Porous Media](#)
- [Theory for Heat Transfer in Solids](#)



The [Thermoelastic Damping](#) subnode is available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.



Heat Generation in a Disc Brake: Application Library path
Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Solid

Heat Transfer in Solids > Solid

Heat Transfer in Fluids > Solid

Heat Transfer in Porous Media > Solid

Heat Transfer in Building Materials > Solid

Bioheat Transfer > Solid

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, or **Bioheat Transfer** selected:

Domains > interface > Solid

Solid with Translational Motion

This node is similar to the **Solid** node, with an additional **Translational Motion** node to account for convection. The **Solid with Translational Motion** node can be used to model rectilinear scrolling motions for block geometries, or rotating motions for cylindrical geometries in stationary or transient analyses. While **Solid** is solved on the material frame, **Solid with Translational Motion** is solved on the spatial frame. It induces that accounting for convection requires a **Moving Mesh** node for **Solid**, while it requires a **Translational Motion** node for **Solid with Translational Motion**. Both methods are exclusive to keep consistent material properties, so it is not possible to combine a **Translational Motion** node with a **Moving Mesh** node. It is recommended to use a **Solid** node and a **Moving Mesh** node to model motions and deformations of solids.



Solid

Translational Motion

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** checkbox 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** checkbox is selected.

Select a **Thermal conductivity** k_d — **From material** (default) or **User defined**, to be used for damaged tissue. For **User defined** choose **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity and enter another value or expression in the field or matrix.

When the material and spatial frames differ (due to the presence in the model of a **Moving Mesh** node, or a Solid Mechanics interface for example), you can select on which frame the **Thermal conductivity** k_d is specified.

By default the **Deformation model for thermal conductivity** is set to **Standard**. With this option, the thermal conductivity is supposed to be given on the material frame. If the material frame does not coincide with the spatial frame, a conversion is applied to get the variables $ht.k_{dxx}$, $ht.k_{dyy}$, and so on. This option is often suitable for moderate elastic strains.

By selecting the **Large strain** option, the thermal conductivity is supposed to be given on the spatial frame. In case of isotropic materials, the thermal conductivity variables $ht.k_{dxx}$, $ht.k_{dyy}$, and so on, are directly equal to the values you have set. In case of anisotropic materials, the rotation of the material is also taken into account following

$$k_{d(x,y,z)} = \mathbf{R}k_{d(X,Y,Z)}\mathbf{R}^T$$

where $\mathbf{R}$ is the rotation matrix between the material and the spatial frames.

THERMODYNAMICS

This section is available when the **Specify different material properties for damaged tissue** checkbox 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.

DISCRETIZATION

This section is available when the **Transformation model** is set to **Temperature threshold** or **Arrhenius kinetics**. To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog. You can change the solution algorithm by using the **Use local time integration** checkbox (cleared by default). When cleared, the solution of the equations for the auxiliary dependent variables in Time Dependent study will be governed by the time stepper algorithm used in the solver.

	• 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 <i>COMSOL Multiphysics Reference Manual</i> .

	<i>Hepatic Tumor Ablation</i>: Application Library path Heat_Transfer_Module/Medical_Technology/tumor_ablation <i>Microwave Heating of a Cancer Tumor</i>: Application Library path Heat_Transfer_Module/Medical_Technology/microwave_cancer_therapy <i>Modeling a Conical Dielectric Probe for Skin Cancer Diagnosis</i>: 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

$$\lambda_{ijkl} = \lambda_{tr} \delta_{ij} \delta_{kl} + \frac{\lambda_{10} - \lambda_{tr}}{2} (\delta_{ik} \delta_{jl} + \delta_{il} \delta_{jk})$$

When the **Horizontal and vertical transverse dispersivities** option is selected in the **Define transverse dispersivity from** list, the **Transverse horizontal dispersivity** $\lambda_{tr, h}$ and **Transverse vertical dispersivity** $\lambda_{tr, v}$ are defined instead of the **Transverse dispersivity** λ_{tr} .

In this case, it is assumed that z is the vertical direction and it defines the dispersion tensor as

$$D_{ij} =$$

$$\frac{1}{|\mathbf{u}|} \begin{bmatrix} \lambda_{10} u^2 + \lambda_{tr, h} v^2 + \lambda_{tr, v} w^2 & (\lambda_{10} - \lambda_{tr, h}) uv & (\lambda_{10} - \lambda_{tr, v}) uw \\ (\lambda_{10} - \lambda_{tr, h}) uv & \lambda_{tr, h} u^2 + \lambda_{10} v^2 + \lambda_{tr, v} w^2 & (\lambda_{10} - \lambda_{tr, v}) vw \\ (\lambda_{10} - \lambda_{tr, v}) uw & (\lambda_{10} - \lambda_{tr, v}) vw & \lambda_{tr, v} u^2 + \lambda_{tr, v} v^2 + \lambda_{10} w^2 \end{bmatrix}$$



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

$$Q_{ted} = -\alpha T : \frac{\partial S}{\partial t}$$

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:

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 4-177](#) and [Equation 4-178](#).

THICKNESS

Specify a value for the **Thickness** d_z of the domain in the out-of-plane direction. This value replaces the overall thickness in the domains that are selected in the **Domain Selection** section, and is used to multiply some terms into the heat equation.



The **Thickness** feature is disabled when at least one of the multiphysics couplings ([Heat Transfer with Surface-to-Surface Radiation](#), [Heat Transfer with Radiation in Participating Media](#), [Heat Transfer with Radiative Beam in Absorbing Media](#)) is active while solving on the same domain. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.



[Out-of-Plane Heat Transfer](#)

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Thickness

Heat Transfer in Solids > Thickness

Heat Transfer in Fluids > Thickness

Heat Transfer in Porous Media > Thickness

Heat Transfer in Building Materials > Thickness

Bioheat Transfer > Thickness

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 6-15, defined in the parent node:

$$-\rho C_p \mathbf{u}_{\text{trans}} \cdot \nabla T$$

The contribution describes the effect of a moving coordinate system, which is required to model, for example, a moving heat source.



Special care must be taken on boundaries where $\mathbf{n} \cdot \mathbf{u}_{\text{trans}} \neq 0$. The **Heat Flux** boundary condition does not, for example, work on boundaries where $\mathbf{n} \cdot \mathbf{u}_{\text{trans}} < 0$.

DOMAIN SELECTION

By default, the selection is the same as for the **Solid with Translational Motion** 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 with Translational Motion** 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 with Translational Motion



Friction Stir Welding of an Aluminum Plate: Application Library path
Heat_Transfer_Module/Thermal_Contact_and_Friction/friction_stir_welding

LOCATION IN USER INTERFACE

A default **Translational Motion** node is automatically added to the [Solid with Translational Motion](#) node.

Context Menus

Heat Transfer in Solids > Solid with Translational Motion > Translational Motion

More locations are available. For example:

Heat Transfer in Fluids > Solid with Translational Motion > Translational Motion

Ribbon

Physics tab with **Solid with Translational Motion** 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.

For flows in porous media, viscous dissipation additionally contains the Darcy term, and the Forchheimer drag term. It is defined by

$$Q_{vd} = \tau : \nabla \mathbf{u} + \frac{\mu}{\kappa} \mathbf{u} \cdot \mathbf{u} + \rho \frac{c_F}{\sqrt{\kappa}} |\mathbf{u}|^3$$

where $\mathbf{u}$ denotes in this case the Darcy velocity, κ is the permeability, and c_F is the Forchheimer parameter.

	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 checkbox under the Energy Balance section. In this case, the Viscous Dissipation feature is overridden by the multiphysics coupling node's contribution.
	• Fluid • Porous Medium • Phase Change Material
	This feature was previously called Viscous Heating.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Fluids > Fluid > Viscous Dissipation

Heat Transfer in Fluids > Porous Medium > Viscous Dissipation

Heat Transfer in Moist Air > Moist Air > Viscous Dissipation

Heat Transfer in Porous Media > Porous Medium > Fluid > Viscous Dissipation

Ribbon

Physics tab with **Fluid**, **Moist Air**, **Porous Medium**, or **Porous Medium > Fluid** selected in the model tree:

Attributes > Viscous Dissipation

Boundary Features

The following boundary nodes and subnodes are available for the Heat Transfer and Heat Transfer in Shells interfaces:

- Boundary Heat Source
- Boundary Pair
- Continuity
- Deposited Beam Power
- External Temperature (Thin Layer, Thin Film, Fracture)
- Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface)
- Harmonic Perturbation
- Heat Flux
- Heat Source (Heat Transfer in Shells Interface)
- Heat Source (Thin Layer, Thin Film, Fracture)
- Inflow
- Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Isothermal Domain Interface
- Line Heat Source on Axis
- Local Thermal Nonequilibrium Boundary
- Lumped System Connector
- Open Boundary
- Outflow
- Periodic Condition (Heat Transfer Interface)
- Phase Change Interface
- Phase Change Interface, Exterior
- Surface-to-Ambient Radiation (Heat Transfer Interface)
- Symmetry (Heat Transfer Interface)
- Temperature
- Thermal Contact
- Thermal Insulation
- Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)
- Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface)



For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries into account and automatically adds an **Axial Symmetry** node that is valid on the axial symmetry boundaries only.

The following boundary nodes and subnodes are available for the Surface-to-Surface Radiation, Radiation in Participating Media, Radiation in Absorbing-Scattering Media, and Radiative Beam in Absorbing Media interfaces:

- Continuity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Continuity on Interior Boundary (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Diffuse Mirror (Surface-to-Surface Radiation Interface)
- Diffuse Surface (Surface-to-Surface Radiation Interface)
- Incident Intensity (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Incident Intensity (Radiative Beam in Absorbing Medium Interface)
- Initial Values (Surface-to-Surface Radiation Interface)
- Layer Opacity (Surface-to-Surface Radiation Interface)
- Opaque Surface (Surface-to-Surface Radiation Interface)
- Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Opaque Surface (Radiative Beam in Absorbing Medium Interface)
- Periodic Condition (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Prescribed Radiosity (Surface-to-Surface Radiation Interface)
- Radiation Group (Surface-to-Surface Radiation Interface)
- Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Semitransparent Surface (Surface-to-Surface Radiation Interface)
- Specular Mirror (Surface-to-Surface Radiation Interface)
- Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)
- Transparent Surface (Radiative Beam in Absorbing Medium Interface)



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

Boundary Heat Source

This node models a heat source (or heat sink) that is embedded in the boundary. When selected as a **Boundary Heat Source**, it also prescribes that the temperature field is continuous across the pair. Note that in this case the source term is applied on the source side.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must be created first. See [Identity, Contact, and Sector Symmetry Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Boundary Heat Source** section are defined in the material or spatial frame:

- The default option for the **Boundary Heat Source** node is **Solid**, which specifies that the heat source Q_b is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See [Conversion Between Material and Spatial Frames](#) for details.
- The **Nonsolid** option defines Q_b in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.



This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See [Handling Frames in Heat Transfer](#) and [Material and Spatial Frames](#) for details.

BOUNDARY HEAT SOURCE

Select an option from the **Heat Source** list:

- For **General source** (default) enter a value for the boundary heat source Q_b when the default option, **User defined**, is selected.
- A positive Q_b corresponds to heating and a negative Q_b corresponds to cooling. For the general boundary heat source Q_b , there are predefined heat sources available when simulating heat transfer together with electrical or electromagnetic interfaces. Such sources represent, for example, ohmic heating and induction heating.



When **User defined** is selected for **General source**, if the expression contains a variable that is discontinuous across that boundary (a thin resistive layer, for example), its value is the average of the value on the “up” and “down” sides of the boundary. You can use the up and down operators to get the value on either side of the boundary (see [up and down](#)).

- 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.

CONTINUITY

If this node is selected from the **Pairs** menu, select the **Prescribe continuity** checkbox to prescribe that the temperature field is continuous across the pair where the source and destination boundaries match. The continuity on a pair should be set from a single feature. It may also be set by a **Continuity** or another **Boundary Heat Source** pair feature.

When the **General source** option is selected, the heat source Q_b is applied on the source or on the destination or on both depending on the selected option in **Source position**. When the **Prescribe continuity** checkbox is selected in this feature (the status in other **Boundary Heat Source** features has no effect) and **General source** is set to **User defined**, $Q_b/2$ is applied on the source and on the destination.

When the **Heat rate** option is selected, the heat rate is distributed on the source or on the destination or on both depending on the selected option in **Source position**. In this case, the integral of the applied heat source is equal to the heat rate.

	To define the boundary heat source Q_b as a function of the temperature, use the local temperature variable on the selected boundary, <code>ht.bhs1.Tvar</code>, that corresponds to the appropriate variable (upside, downside, or average temperature of a layer, wall temperature with turbulence modeling), depending on the model configurations. See Boundary Wall Temperature for a thorough description of the boundary temperature variables.
	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose Harmonic Perturbation. Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.
	Upside and downside settings can be visualized by plotting the global normal vector (<code>nx</code>, <code>ny</code>, <code>nz</code>), that always points from downside to upside. Note that the normal vector (<code>ht.nx</code>, <code>ht.ny</code>, <code>ht.nz</code>) may be oriented differently. See Tangent and Normal Variables in the <i>COMSOL Multiphysics Reference Manual</i>.
	• Handling Frames in Heat Transfer • About the Heat Transfer Interfaces
	<i>Evaporative Cooling of Water</i>: Application Library path Heat_Transfer_Module/Phase_Change/evaporative_cooling



When **Line Heat Flux** is applied on a pair, the flux is only applied on the edge adjacent to the source boundary which is in contact with the destination boundary. Consider adding another pair with opposite source and destination boundaries to apply a flux on the edge adjacent to the destination boundary and in contact with the source boundary.

See [Identity, Contact, and Sector Symmetry Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Heat Sources > Boundary Heat Source

Heat Transfer in Solids > Pairs > Boundary Heat Source

Heat Transfer in Porous Media > Porous Medium > Fluid > Heat Sources > Boundary Heat Source

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Heat Sources > Boundary Heat Source

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries > interface > Boundary Heat Source

Pairs > interface > Boundary Heat Source

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Boundary Heat Source

Boundary Pair

Use the **Boundary Pair** node to set the periodicity conditions over the boundaries of the **Cell Periodicity** domain. This node is only available when the **Boundary condition type** in the **Cell Periodicity** parent feature is set to **Periodic temperature**. In 2D, at least two such pairs are needed, and in 3D at least three pairs. The number of pairs depends on the shape of the unit cell.

Select all corresponding boundaries belonging to both sides of the pair. The source and destination sides of the pairs can be manually specified by adding a **Manual Destination Selection** subnode.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default), **Weak constraints** or **Nitsche constraints**.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Solids and Fluids > **Specific Media** > **Cell Periodicity** > **Boundary Pair**
- Heat Transfer in Solids > **Specific Media** > **Cell Periodicity** > **Boundary Pair**
- Heat Transfer in Fluids > **Specific Media** > **Cell Periodicity** > **Boundary Pair**
- Heat Transfer in Porous Media > **Specific Media** > **Cell Periodicity** > **Boundary Pair**
- Heat Transfer in Building Materials > **Specific Media** > **Cell Periodicity** > **Boundary Pair**
- Bioheat Transfer > **Specific Media** > **Cell Periodicity** > **Boundary Pair**

Ribbon

Physics tab with **Cell Periodicity** node selected in the model tree:

Attributes > Boundary Pair

Continuity

This node can be added to pairs, or as a subnode of the **Fluid** and **Porous Matrix** subnodes under **Porous Medium**. It prescribes that the temperature field is continuous across the boundary. When applied to pairs, **Continuity** is only suitable for pairs where the boundaries match. When added as a subnode of the **Fluid** and **Porous Matrix** subnodes under **Porous Medium**, it ensures the continuity of the temperature and flux between the specific phase (fluid or solid) and the medium on the other side of the boundary (fluid, solid, building material, biological tissue, ...). If a thin structure

model ([Thin Layer \(Heat Transfer Interface\)](#) and [Solid \(Heat Transfer in Shells Interface\)](#), [Thin Film \(Heat Transfer Interface\)](#) and [Fluid \(Heat Transfer in Shells Interface\)](#), or [Fracture \(Heat Transfer Interface\)](#) and [Porous Medium \(Heat Transfer in Shells Interface\)](#))) is defined on the boundary, the **Continuity** node may be applied as well.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must be created first. See [Identity, Contact, and Sector Symmetry Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	<i>Thermomechanical Analysis of a Surface-Mounted Resistor:</i> Application Library path Heat_Transfer_Module/Thermal_Stress/surface_resistor

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Solids and Fluids > Pairs > Continuity**
- Heat Transfer in Shells > Pairs > Continuity**
- Surface-to-Surface Radiation > Pairs > Continuity**
- Radiative Beam in Absorbing Media > Pairs > Continuity**
- Heat Transfer in Porous Media > Porous Medium > Fluid > Continuity**
- Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Continuity**

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface, **Heat Transfer in Shells**, **Radiative Beam in Absorbing 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 be created first. See [Identity](#), [Contact](#), and [Sector Symmetry Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.



- [Constraint Reaction Terms](#)
 - [Weak Constraints](#)
 - [Constraint Settings](#)
-

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 **Beam power density**, I_0 (SI unit: W/m^2). The **Beam power density** corresponds to the beam intensity in the beam cross section.

For **Built-in beam profiles**, enter a value for the **Deposited beam power P_0** and the coordinates of the **Beam origin point $\mathbf{O}$** .

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 <i>COMSOL Multiphysics Reference Manual</i> .

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Heat Sources > Deposited Beam Power
Heat Transfer in Porous Media > Porous Medium > Fluid > Heat Sources > Deposited Beam Power
Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Heat Sources > Deposited Beam Power

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries > *interface* > **Deposited Beam Power**

Physics tab with **Porous Medium** > **Fluid** or **Porous Medium** > **Porous Matrix** selected in the model tree:

Attributes > **Deposited Beam Power**

Diffuse Mirror (Surface-to-Surface Radiation Interface)

This node is a variant of the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node with a surface emissivity equal to zero. Diffuse mirror surfaces are common as approximations of a surface that is well insulated on one side and for which convection effects can be neglected on the opposite (radiating) side. It resembles a mirror that absorbs all irradiation and then radiates it back in all directions. The node adds radiosity shape function for each spectral band to its selection and uses it as surface radiosity.

The radiative heat flux on a diffuse mirror boundary is zero.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when temperature-dependent material properties are used. The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

RADIATION DIRECTION

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

AMBIENT

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

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 pure specular reflection should be considered, use the [Specular Mirror \(Surface-to-Surface Radiation Interface\)](#) node instead.

If specular reflection should be considered, use the [Opaque Surface \(Surface-to-Surface Radiation Interface\)](#) node instead.

If specular reflection and transmission should be considered, use the [Semitransparent Surface \(Surface-to-Surface Radiation Interface\)](#) node instead.

If no emission should be considered, use the [Diffuse Mirror \(Surface-to-Surface Radiation Interface\)](#) node instead.

MODEL INPUT

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups have been added, the model inputs are included here.

There is one standard model input: the **Temperature** T is used in the expression of the blackbody radiation intensity and when multiple wavelength intervals are used, for the fractional emissive power. The temperature model input is also used to determine the variable that receives the radiative heat source. When the model input does not contain a dependent variable, the radiative heat source is ignored.

The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

RADIATION DIRECTION

The radiation directions are shown on the geometry in the **Graphics** window. A symbol () is displayed on boundaries that do not radiate. Hide the symbols with the **Show Radiation Direction** button () (selected by default).

When **Wavelength dependence of radiative properties** is set to **Constant** in the **Radiation Settings** section of the physics interface, select an **Emitted radiation direction** based on the geometric normal (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 \(Surface-to-Surface Radiation and Orbital Thermal Loads Interfaces\)](#) domain feature. For external boundaries, the exterior side opacity is transparent by default but may be edited by setting the **Selection** of the **Opacity** feature on **All voids** in the **Opacity** feature. [Figure 6-1](#)

illustrates the emitted radiation direction depending on the opacity of the adjacent domain.

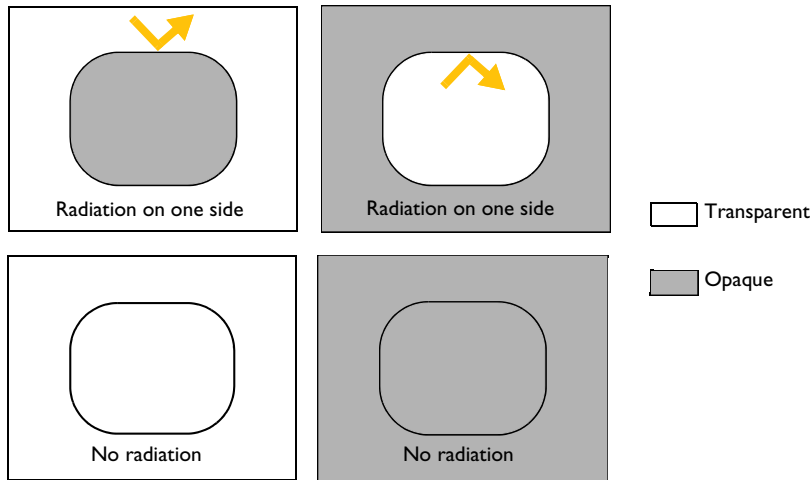


Figure 6-1: Emitted radiation direction with the option Opacity controlled.

- Select **Negative normal direction** to specify that the surface radiates in the negative normal direction (**un** vector direction). An arrow indicates the negative normal direction that corresponds to the direction of the radiation emitted by the surface. [Figure 6-2](#) illustrates the emitted radiation direction depending on the chosen option.
- Select **Positive normal direction** if the surface radiates in the positive normal direction (**dn** vector direction). An arrow indicates the positive normal direction that corresponds to the direction of the radiation emitted by the surface. [Figure 6-2](#) illustrates the emitted radiation direction depending on the chosen option.

- Select **Both sides** if the surface radiates on both sides. [Figure 6-2](#) illustrates the emitted radiation direction depending on the chosen option.
- Select **None** when adjacent domains are either both transparent or both opaque for a given spectral band. When the **Emitted radiation direction** is set to **None** for a spectral band, the radiative properties set for this spectral band are not used

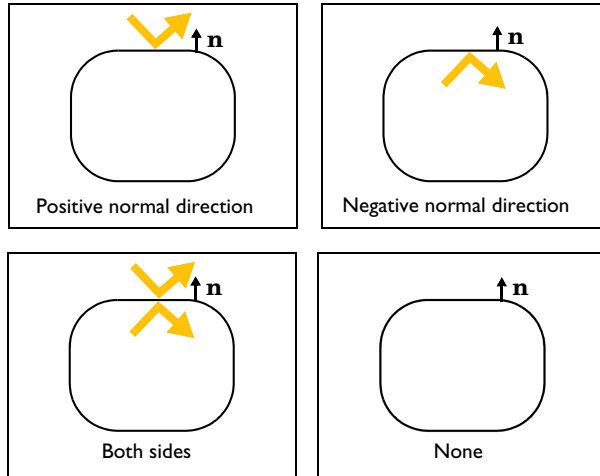


Figure 6-2: Emitted radiation direction with Positive normal direction, Negative normal direction and Both sides options.

When **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands** in the **Radiation Settings** section of the physics interface:

- When **Emitted radiation direction** is set to **User defined for all bands** (default), select a radiation direction that is the same for all spectral bands: **Opacity controlled** (default), **Negative normal direction**, **Positive normal direction**, **Both sides**, or **None**. Use this option if the transparency or opacity properties of the material are not wavelength dependent.
- When **Emitted radiation direction** is set to **User defined for each band**, select a radiation direction for each spectral band: **Opacity controlled** (default), **Negative normal direction**, **Positive normal direction**, **Both sides**, or **None**. The **Emitted radiation direction** defines the radiation direction for each spectral band similarly as when **Wavelength dependence of radiative properties** is **Constant**. Defining a radiation

direction for each spectral band makes it possible to build models where the transparency or opacity properties defers between spectral bands.



This is useful for example to represent glass opaque to radiation outside of the $[0.3\text{--}2.5\text{ }\mu\text{m}]$ wavelength range.



Note that when **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, the upper bound of the last spectral band, meant to represent the infinite, is set to 1 [mm] , for the computation of the surface material properties.

AMBIENT

Select the **Define ambient properties on each side** checkbox when the ambient properties differs between the sides of a boundary. This is needed to define ambient temperature for a surface that radiates on both sides and that is exposed to a hot temperature on one side (for example, fire) and to a cold temperature on the other side (for example, external temperature). By default, **Define ambient properties on each side** is not selected.

Ambient Temperature

The ambient temperature T_{amb} should be set for the computation of $e_b(T_{\text{amb}})$, the ambient blackbody hemispherical total emissive power, used for the evaluation of the ambient irradiation $G_{\text{amb}} = F_{\text{amb}}\epsilon_{\text{amb}}e_b(T_{\text{amb}})$.

Set the **Ambient temperature** T_{amb} . For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. When **Define ambient properties on each side** is selected, define the **Ambient temperature, upside** $T_{\text{amb,u}}$ and **Ambient temperature, downside** $T_{\text{amb,d}}$, respectively. The geometric normal points from the downside to the upside.



Set T_{amb} to the far-away temperature in directions where no other boundaries obstruct the view. Inside a closed cavity, the ambient view factor, F_{amb} , is theoretically zero and the value of T_{amb} therefore should not matter. It is, however, good practice to set T_{amb} to T or to a typical temperature value for the cavity surfaces in such cases because that minimizes errors introduced by the finite resolution of the view factor evaluation.

Ambient Emissivity

The ambient emissivity ϵ_{amb} should be set for the computation of the ambient irradiation $G_{\text{amb}} = F_{\text{amb}} \epsilon_{\text{amb}} e_b(T_{\text{amb}})$.

The **Ambient emissivity** ϵ_{amb} should be specified. When **Blackbody** is selected, the ambient emissivity is set to 1 for the computation of the ambient irradiation G_{amb} .

If **Wavelength dependence of radiative properties** is **Constant**, choose alternatively the **User defined** option, and set a value or expression. You can define a temperature-dependent emissivity using the variable rad.T .

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the ambient irradiation $G_{\text{amb},i}$ 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 properties 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

When the **Include diffuse irradiance** checkbox is selected, a diffuse irradiation contribution I_{diff} is included into the external irradiation. When considering solar irradiation, it takes into account the irradiation from the Sun, scattered by the atmosphere, and assumed to be isotropic.

- When **Diffuse irradiance** is set to **User defined**, enter a value or expression for the diffuse irradiance.
- When **Diffuse irradiance** is set to **User defined for each band**, enter a value or expression for the diffuse irradiance for each spectral band.
- Select a **Clear sky noon diffuse horizontal irradiance** defined in an **Ambient Properties** node under **Definitions** to define the diffuse irradiance from ambient properties.

To consider only the direct irradiation defined in the [External Radiation Source](#) feature, clear the **Include diffuse irradiance** checkbox.

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** checkbox 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 **Emissivity** ϵ (dimensionless) uses values **From material**. This is a property of the material surface that depends both on the material itself and the structure of the surface. Make sure that a material is defined at the boundary level (by default materials are defined at the domain level).
- For **User defined**, set a value or expression. You can define a temperature-dependent emissivity using the variable `rad.T`.

Select the **Define properties on each side** checkbox to set specific values for the radiative properties on each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside** or **Emissivity, downside** are **From material**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Emissivity** ε (dimensionless) uses values **From material**.
- When **Emissivity** is set to **User defined**, enter a value or expression for the **Emissivity** ε . The wavelength may be accessed through the `rad.lambda` variable. Any expression set for the emissivity is then averaged on each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.
- When **Emissivity** is set to **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is defined on both sides. Select the **Define properties on each side** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

Set the surface emissivity to a number between 0 and 1, where 0 represents diffuse mirror and 1 is appropriate for a perfect blackbody. The proper value for a physical material lies somewhere in between and can be found from tables or measurements.

	In the notation used here, Bi stands for B1 , B2 , ... up to the maximum number of spectral intervals.
	To define temperature dependencies for the user inputs (surface emissivity for example), use the temperature variable <code>ht.T</code> , that corresponds to the appropriate variable (upside, downside, or average temperature of a layer, wall temperature with turbulence modeling), depending on the model configurations. See Boundary Wall Temperature for a thorough description of the boundary temperature variables.
	Several settings for this node depend on the Wavelength dependence of radiative properties setting, which is defined for the physics interface. In addition, the Transparent media refractive index is equal to 1 by default. See Radiation Settings .

	Theory for Surface-to-Surface Radiation
	Upside and downside settings can be visualized by plotting the global normal vector (nx, ny, nz), that always points from downside to upside. Note that the normal vector (ht.nx, ht.ny, ht.nz) may be oriented differently. See Tangent and Normal Variables in the <i>COMSOL Multiphysics Reference Manual</i>.
	<i>Heat Generation in a Disc Brake</i>: 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** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	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

For a manually added **Porous Medium** node in the Heat Transfer in Fractures interface, or for a **Fracture** node in the Heat Transfer interface, select the **Restrict to layered boundaries** checkbox 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.

The default **Porous Medium** node of the Heat Transfer in Fractures interface 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** checkbox 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** checkbox 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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes for the selection of the individual layers.
- Alternatively, set **Shell type** to **Nonlayered shell**, and set a user-defined value or expression for the **Thickness** L_{th} . This option should be used for thermally thin layers, for which no through-thickness temperature variation is expected in the layered material. This removes the **General** option from the **Layer type** list in the **Layer Model** section.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.



This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include enthalpy (the energy flux, for example).

It is also used if the ideal gas law is applied. See [Thermodynamics, Fluid](#).

The default **Absolute pressure** p_A is **User defined**. When additional physics interfaces are added to the model, the absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Laminar Flow** interface is added you

can select **Absolute pressure (spf)** from the list. The **Common model input** option corresponds to the `minput.pA` variable, set to 1 atm by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section.

LAYER MODEL

The available options for **Layer type** are **Thermally thin approximation** and **General**. The former is a lumped model that accounts only for tangential temperature gradients (along the fracture), whereas the latter accounts also for the normal gradients of temperature (through the fracture’s thickness).

	Within a layered material selection, a single Layer type should be used. If two layer types are needed for the same layered material, the original material should be duplicated so that one layered material is defined for each layer type. A Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface) node may be added between the two layered materials.

FLUID MATERIAL

By default, the **Boundary material** is used.

HEAT CONDUCTION, FLUID

The settings are the same as for [Thin Film \(Heat Transfer Interface\)](#) and [Fluid \(Heat Transfer in Shells Interface\)](#).

HEAT CONDUCTION, FLUID

The settings are the same as for [Thin Film \(Heat Transfer Interface\)](#) and [Fluid \(Heat Transfer in Shells Interface\)](#).

THERMODYNAMICS, FLUID

The settings are the same as for [Thin Film \(Heat Transfer Interface\)](#) and [Fluid \(Heat Transfer in Shells Interface\)](#).

POROUS MATERIAL

By default, the **Boundary material** is used. The **Volume fraction** θ_p should be specified. The default is 0.

HEAT CONDUCTION, POROUS MATRIX

The default **Thermal conductivity** k_p is taken **From layered material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** based on the characteristics of the thermal conductivity, and enter another value or expression. Select an **Effective conductivity**: **Plane layers parallel to heat flow** (default) or **Power law**.

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 > interface > Fracture

Physics tab with *interface* as **Heat Transfer in Shells**, **Heat Transfer in Films**, or **Heat Transfer in Fractures** selected:

Boundaries > interface > Porous Medium

Harmonic Perturbation

Use this subnode to specify the harmonic variation of a prescribed temperature value. It is used only in a study step of type **Frequency-Domain Perturbation**.

Enter a value or expression for the **Temperature perturbation** ΔT_0 to be added to the equilibrium temperature T_0 defined in the parent node.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids > Temperature > Harmonic Perturbation

More locations are available. For example:

Heat Transfer in Solids > Thin Rod > Temperature > Harmonic Perturbation

Heat Transfer in Solids > Thin Layer > Temperature > Harmonic Perturbation

Heat Transfer in Solids > Thin Film > Temperature > Harmonic Perturbation

Heat Transfer in Solids > Fracture > Temperature > Harmonic Perturbation

Heat Transfer in Solids > Thin Layer > External Temperature > Harmonic Perturbation

Heat Transfer in Shells > Temperature > Harmonic Perturbation

Ribbon

Physics tab with **Temperature** or **External Temperature** selected in the model tree:

Attributes > Harmonic Perturbation

Heat Flux

Use this node to add heat flux across boundaries. A positive heat flux adds heat to the domain. This feature is not applicable to inlet boundaries, use the [Inflow](#) condition instead.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat flux q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See [Conversion Between Material and Spatial Frames](#) for details.

- The default option for the **Heat Flux** node is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.



This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See [Handling Frames in Heat Transfer](#) and [Material and Spatial Frames](#) for details.

HEAT FLUX

Select the **Flux type** from the list: **General inward heat flux** (default), **Convective heat flux**, **Heat rate**, or **Nucleate boiling heat flux**.

General Inward Heat Flux

It adds q_0 to the total flux across the selected boundaries. Enter a value for q_0 to represent a heat flux that enters the domain. For example, any electric heater is well represented by this condition, and its geometry can be omitted.

Convective Heat Flux

The default option is to enter a **User defined** value for the **Heat transfer coefficient** h .

In addition, the following options are also available to control the type of convective heat flux to model: **External natural convection**, **Internal natural convection**, **External forced convection**, or **Internal forced convection**.

For all options except **User defined**, select a **Fluid**: **Air** (default), **Transformer oil**, **Water**, **Moist air**, or **From material**.

When **From material** is selected, choose a material available on the boundary from the **Materials** list.

Depending of the selected option, different parameters are needed. You can refer to the **Sketch** section to get an illustration of the configuration.

External Natural Convection

For **External natural convection** select **Vertical wall**, **Inclined wall**, **Horizontal plate, upside**, **Horizontal plate, downside**, **Long horizontal cylinder**, **Sphere**, or **Vertical Thin Cylinder** from the list under **Heat transfer coefficient**. Then enter the applicable information:

- **Wall height L** .
- **Wall height L** and the **Tilt angle ϕ** . The tilt angle is the angle between the wall and the vertical direction, $\phi = 0$ for vertical walls.
- **Characteristic length (area/perimeter) L** . The characteristic length is the ratio between the surface area and its perimeter.
- **Cylinder diameter D** .
- **Sphere diameter D** .
- **Cylinder height H** .

Internal Natural Convection

For **Internal natural convection** select **Narrow chimney, parallel plates** or **Narrow chimney, circular tube** from the list under **Heat transfer coefficient**. Then enter the applicable information:

- **Plate distance L** and a **Chimney height H** .
- **Tube diameter D** and a **Chimney height H** .

External Forced Convection

For **External forced convection** select **Plate, averaged transfer coefficient**, **Plate, local transfer coefficient**, **Cylinder in cross flow**, or **Sphere** from the list under **Heat transfer coefficient**. Then enter the applicable information:

- **Plate length L** and **Velocity, fluid U** .
- **Position along the plate x_{pl}** and **Velocity, fluid U** .
- **Cylinder Diameter D** and **Velocity, fluid U** .
- **Sphere Diameter D** and **Velocity, fluid U** .

Internal Forced Convection

For **Internal forced convection** the only option is **Isothermal tube**. Enter a **Tube diameter D** and a **Velocity, fluid U** .

If **Velocity, fluid U** is **User defined**, enter a value or expression. Else, select a **Wind velocity** defined in the [Consistent Stabilization](#) section of a Heat Transfer or Heat Transfer in Shells interface.

External conditions

First, set the **Absolute pressure**, p_A . For **User Defined**, enter a value or expression. Else, select an **Ambient absolute pressure** defined in an **Ambient Properties** node under **Definitions**. The pressure is used to evaluate the **Fluid** material properties and this setting is not available for the **Transformer oil** and **Water** options.

In addition, enter an **External temperature**, T_{ext} . For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

Finally, when the **Fluid** is **Moist air**, also set the **External relative humidity**, $\phi_{w,\text{ext}}$, 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.

Nucleate Boiling Heat Flux

This option computes q_0 with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See [Nucleate Pool Boiling Correlation](#) for details about the correlation.

The correlation applies to clean surfaces, and is insensitive to the shape and orientation of the surface. It relies on empirical constants C_{sf} and s that are predefined for several combinations of fluid and surface materials, and on the boiling fluid properties at saturation temperature, also predefined for some fluids.

Select an option in the **Fluid** list:

- When **Water**, **Benzene**, **n-Pentane**, or **Ethanol** is selected, the material properties of the liquid and vapor phases required in the correlation are predefined (at saturation temperature and $p_A=1$ atm). Depending on the selected fluid, different materials and finishings are available in the **Surface** list. For each option, the empirical constants C_{sf} and s are predefined.
- When **From material** is selected in the **Fluid** list, first choose materials available on the boundary from the **Liquid materials** and **Vapor materials** lists, to define ρ_l , $C_{p,l}$, k_l , μ_l , and ρ_v . Then set the **Ambient pressure**, p_A , the **Saturation temperature**, T_{sat} , the **Latent heat of evaporation**, L_v , and the **Liquid-vapor surface tension**, σ . Finally, enter values or expressions for the Rohsenow's correlation parameters: the **Prandtl number exponent**, s , and the **Liquid-surface combination factor**, C_{sf} , which accounts for

surface roughness, that tends to increase the number of active nucleation sites for boiling.

	This boundary condition will give incorrect results if $T < T_{\text{sat}}$ and should only be used in a limited interval of validity $T_{\text{sat}} + T_1 < T < T_{\text{sat}} + T_2$. Typically, for water $T_1=10$ K and $T_2=30$ K.
	• The detailed definition of the predefined heat transfer coefficients for convective heat transfer is given in Convective Heat Transfer Correlations. • For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 21.
	• Handling Frames in Heat Transfer • About the Heat Transfer Interfaces
	• <i>Power Transistor</i>: Application Library path Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/power_transistor • <i>Free Convection in a Water Glass</i>: Application Library path Heat_Transfer_Module/Tutorials_Forced_and_Natural_Convection/cold_water_glass

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Heat Flux

Heat Transfer in Porous Media > Porous Medium > Fluid > Heat Flux

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Heat Flux

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries > interface > Heat Flux

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

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 4-73](#). 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** checkbox 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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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 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 <i>COMSOL Multiphysics Reference Manual</i>.

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** checkbox. For a given **Layered Material Link**, **Layered Material Stack**, or **Single Layer Material**, you get access to a list of checkboxes for the selection of the individual layers.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent node, the **Shell type** is taken **From parent feature** in the **Heat Source** node. You can change to **Nonlayered shell** and override the parent node setting with a specific value or expression for the **Thickness** L_{th} .

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.

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 be created first. See [Identity, Contact, and Sector Symmetry Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

INCIDENT INTENSITY

When **Wavelength dependence of radiative properties** is **Constant**, the **External radiation intensity** I_i should be specified. This represents the value of radiative intensity along incoming discrete directions. Values of radiative intensity on outgoing discrete directions are not prescribed.

If **External radiation intensity** is **User defined**, set a value or expression for I_{ext} .

Alternatively, choose **Blackbody radiation** or **Graybody radiation** to define I_i as $I_b(T_{\text{ext}})$ or $\epsilon I_b(T_{\text{ext}})$, respectively (available with the Radiation in Participating Media interface only). Set the **External temperature** T_{ext} (SI unit: K, default value 293.15 K) to specify the temperature at which the blackbody radiation intensity I_b is evaluated.

Alternatively, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. For **Graybody radiation**, the **Emissivity** ϵ and **External Temperature**

T_{ext} should be defined. If the **Emissivity** is defined **From material**, make sure that a material is defined at the boundary level (materials are defined by default at the domain level). If **User defined** is selected for the **Emissivity**, enter another value for ϵ . If the **Define properties on each side** checkbox is selected, define the **Emissivity, upside** ϵ_u and the **Emissivity, downside** ϵ_d . Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside** or **Emissivity, downside** are **From material**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the **External radiation intensity** $I_{i,k}$ for each spectral band should be specified. Select an option between **Graybody radiation**, **Blackbody radiation**, **User-defined for each band**, or **User-defined distribution**.

If **External radiation intensity** is **Blackbody radiation** or **Graybody radiation** (available with the Radiation in Participating Media interface only), $I_{i,k}$ is defined from the blackbody intensity $I_b(T_{\text{ext}})$, the fractional emissive power for each spectral band at external temperature $\text{FEP}_k(T_{\text{ext}})$, the emissivity ϵ_k and the **External temperature** T_{ext} (SI unit: K, default value 293.15 K). For **Graybody radiation**, the **Emissivity** ϵ_k and **External Temperature** T_{ext} should be defined from a user expression or by selecting an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. For **User-defined for each band**, set a value for each spectral band in the table displayed underneath. Within a spectral band, each value is assumed to be independent of wavelength. If the **Define properties on each side** checkbox is selected, define the **Emissivity, upside** ϵ_u and the **Emissivity, downside** ϵ_d . Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside** or **Emissivity, downside** are **From material**.

If **External radiation intensity** is **User-defined for each band**, set a value or expression for $I_{\text{ext},k}$ for each spectral band in the table displayed underneath. If the **Define incident intensity on each side** checkbox is selected, set values in the **Upside** and **Downside** columns.

If **External radiation intensity** is **User-defined distribution**, set a value or expression for $I_{\lambda,\text{ext}}$. Any expression is then averaged on each spectral band to obtain a piecewise constant radiation intensity. If the average value of the radiation intensity on each band is known, you may use instead the **User-defined for each band** option to avoid the

evaluation of the average. If the **Define incident intensity on each side** checkbox is selected, set values for $I_{\lambda,\text{ext,u}}$ and $I_{\lambda,\text{ext,d}}$.

	When Discrete ordinates method is selected, the components of each discrete ordinate vector can be used in this expression. The syntax is <code>name.sx</code> , <code>name.sy</code> , and <code>name.sz</code> , where <code>name</code> is the physics interface node name. By default, the Radiation in Participating Media interface is <code>rpm</code> so <code>rpm.sx</code> , <code>rpm.sy</code> , and <code>rpm.sz</code> correspond to the components of discrete ordinate vectors.
	When PI approximation is selected, an additional Input quantity setting appears for the User defined , User-defined for each band , and User-defined distribution options. Set it to Radiative intensity to set the external radiation in terms of I , or set it to Incident radiation to set it in terms of G .
	• 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 $\mathbf{e}$** . The orientation vector does not need to be normalized.

BEAM PROFILE

Either select an option for the **Beam profile** among the **Built-in beam profiles**, or set the it as **User defined** by entering a value for the **Beam power density**, I_0 (SI unit: W/m^2). The **Beam power density** corresponds to the beam intensity in the beam cross section.

For **Built-in beam profiles**, enter a value for the **Deposited beam power** P_0 and the coordinates of the **Beam origin point $\mathbf{O}$** .

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.

	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 <i>COMSOL Multiphysics Reference Manual</i> .
	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:

Inflow

Use this node to model inflow of heat through a virtual domain, at inlet boundaries of nonsolid 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:

$$-\mathbf{n} \cdot \mathbf{q} = \rho \Delta H \mathbf{u} \cdot \mathbf{n} \quad (6-16)$$

where ΔH is the sensible enthalpy. See [Theory for the Inflow Boundary Condition](#) for the definition of ΔH and more details on the Danckwerts condition.

UPSTREAM PROPERTIES

The **Upstream temperature** T_{ustr} should be specified. For **User Defined**, enter a value or expression. Else, select any available input (like **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**).

Select the **Specify upstream absolute pressure** checkbox 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 Porous Media > Porous Medium > Fluid > Flow Conditions > Inflow

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Flow Conditions > Inflow

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface selected:

Boundaries > *interface* > **Inflow**

Physics tab with **Porous Medium** > **Fluid** selected in the model tree:

Attributes > **Inflow**

Initial Values (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

This node adds an initial value for the temperature that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver. If more than one set of initial values is needed, add an **Initial Values** node from the **Physics** toolbar.

BOUNDARY SELECTION

When the **Initial Values** node is added manually in the Heat Transfer in Shells interface or under the **Thin Layer**, **Thin Film**, or **Fracture** node, select the **Restrict to layered boundaries** checkbox 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** checkbox 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** checkbox 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 checkbox, 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 checkboxes for the selection of the individual layers.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface or node, the **Thickness** is taken **From physics interface** or **From parent feature** in the **Initial Values** node. This option is not editable when **Initial Values** is the default node or subnode, but you can change to **User defined** and override the parent setting with a specific value or expression for L_{th} when the node was added manually.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.

INITIAL VALUES

For **User defined**, enter a value or expression for the initial value of the **Temperature** T . The default is approximately room temperature, 293.15 K (20° C). Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

When **Initial Values** is added as a subnode of **Thin Layer**, **Thin Film**, or **Fracture** node with **Layer type** set as **Thermally thick approximation**, it is applicable only on exterior boundaries and defines the initial temperature on the outer side of the thin structure. Note that the initial temperature of the inner side of the thin structure is always defined from the initial condition of the adjacent domains.

When **Initial Values** is added as a subnode of **Thin Layer**, **Thin Film**, or **Fracture** node with **Layer type** set as **Thermally thick approximation** or **General**, an additional option, **From adjacent domains**, is available. When it is selected, the initial temperature of the layer is

uniform through the thickness and is equal to the average value of initial temperature in the adjacent domains. Note that this option is not available when the **Initial Values** feature is added in the Heat Transfer in Shells interface.

	Theory for Heat Transfer in Thin Structures
	<i>Disk-Stack Heat Sink</i> : 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** checkbox to set specific values for the **Initial surface radiosity, upside** J_u and **Initial surface radiosity, downside** J_d .
- When **Radiative intensity** is set to **Blackbody/Graybody**, enter a value for the **Initial temperature** T_{init} .

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, set the **Initial value** to **Blackbody/Graybody**, **User defined for each band**, or **User defined**:

- When **Initial value** is set to **User defined**, enter a value or expression for the **Initial surface radiosity distribution** J_λ . The wavelength may be accessed through the `rad.lambda` variable. This distribution is integrated on each spectral band to obtain the source heat rate J_i for each spectral band.
- When **Initial value** is set to **Blackbody/Graybody**, enter a value for the **Initial temperature** T_{init} . For **User defined**, enter a value or expression for the initial value of the **Initial temperature** T_{init} . Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.
- When **Initial value** is set to **User defined for each band**, enter a value for the **Initial surface radiosity** J_i for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same radiosity is defined on both sides. Select the **Define initial surface radiosity on each side** checkbox 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 an **Isothermal domain** feature is added, 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 be created first. See [Identity, Contact, and Sector Symmetry 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**, **Nucleate boiling 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)
Nucleate Boiling Heat Flux	Solid	None	Heat flux (nucleate boiling)
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:

- $\phi_{d \rightarrow u}$ is the mass flux in the positive direction and $\phi_{u \rightarrow d}$ 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. For internal boundaries, the external temperature is defined as the temperature of the adjacent domain.

Nucleate Boiling Heat Flux

The heat flux q_0 is computed with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See [Nucleate Pool Boiling Correlation](#) for details about the correlation. This option is intended to be used on an exterior boundary and considers the **Isothermal Domain** to be the solid. It has no effect on interior boundaries. Select materials from the **Fluid** and **Surface** lists. When a predefined material (**Water**, **Benzene**, **n-Pentane**, or **Ethanol**) is used for the fluid, no additional parameters are required, because the liquid and vapor properties are predefined (at saturation temperature) and the Rohsenow's correlation parameters C_{sf} and s are available for different surface materials. Else, see [Nucleate Boiling Heat Flux](#) ([Heat Flux](#) feature) for details about the settings.

Thermal Contact

This condition specifies the conductive heat flux at the interface of a solid isothermal domain adjacent to a solid.

Either specify the **Surface thermal resistance**, $R_{t,s}$ (SI unit: $K \cdot m^2/W$), or set the **Thermal resistance**, R_t (SI unit: K/W) and evaluate $R_{t,s}$ from it:

$$R_{t,s} = R_t A$$

where A (SI unit: m^2) is the surface area of the boundary.

The **External temperature** found under the section of the same name must be set when the isothermal domain interface is also an exterior boundary.

EXTERNAL TEMPERATURE

This section is not available if the **Interface type** is set to **Thermal Insulation** or **Continuity**. Else, the **External temperature**, T_{ext} should be specified. It is used to calculate the heat exchange with the exterior by the **Ventilation**, **Convective heat flux**, and **Thermal contact** options on isothermal domain interfaces that are also exterior boundaries. The value is ignored on interior boundaries.

For **User defined**, enter a value or expression for the **Temperature** T_{ext} . Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

INTERFACE OPACITY

Select **Opaque** (default) or **Transparent** to set the interface’s opacity type.

This is needed when the interface contributes with any boundary condition from the **Radiation** menu. It picks the side where irradiation starts from.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	• Isothermal Domain • Theory for Lumped Isothermal Domain • The detailed definition of the predefined heat transfer coefficients is given in Convective Heat Transfer Correlations.
	• For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in Ref. 21.



LOCATION IN USER INTERFACE

Context Menus

If an **Isothermal Domain** feature is present:

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** (default) 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**:

- When **Opacity** is set to **User defined for all bands** (default), set the opacity of the layer for all bands by selecting **Opaque** (default) or **Transparent**. Use this option if the transparency or opacity properties of the material are not wavelength dependent.
- When **Opacity** is set to **User defined for each band**, set the opacity for each spectral band by selecting the corresponding checkbox in the **Opaque** column of the table. By default the checkbox is selected for all spectral bands. Defining an opacity for each spectral band makes it possible to build models where the transparency or opacity properties of the material defers between 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

Local Thermal Nonequilibrium Boundary

This node is active by default on all the interior and exterior boundaries of the computational domain, adjacent to domains where a **Porous Medium** node is applied with **Porous medium type** set to **Local thermal nonequilibrium**. It shows which boundaries are adjacent to domains where two temperatures, T_f and T_s , are solved.

On these boundaries, the conditions available at the physics interface level are not applicable, because they define conditions on a single temperature, T . There are however a few exceptions on interior boundaries:

- The thin structures conditions (**Thin Layer**, **Thin Film**, **Fracture**, and **Thermal Contact**), which represent a thin domain as a boundary, where a single temperature, T , is

solved. They can be combined with other boundary conditions on T , available at the physics interface level on boundaries where a thin structure condition is applied:

Boundary Heat Source, **Deposited Beam Power**, and **Surface-to-Ambient Radiation**.

- The **Thermal Insulation** node, which enforces thermal insulation for both the fluid and solid phases.
- The **Periodic Condition** node, which applies a periodic condition for each phase.

Other boundary conditions can be applied specifically to the fluid or porous matrix phase, by adding subnodes to the [Fluid \(Porous Medium\)](#) and [Porous Matrix \(Porous Medium, Moist Porous Medium\)](#) subnodes.

Concerning the domain conditions, the **Heat Source** and **Initial Values** nodes, available at the physics interface level, are not applicable in the domains delimited by the **Local Thermal Nonequilibrium Boundary** condition.

This node is automatically added and removed when necessary.

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. The initial temperature of the connector is also taken from the selected **External Terminal** feature for a transient simulation.

When the **Isothermal terminal** checkbox 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 checkbox to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.



Lumped Composite Thermal Barrier: Application Library path
**Heat_Transfer_Module/Tutorials,_Thin_Structure/
lumped_composite_thermal_barrier**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Lumped System Connector

Heat Transfer in Porous Media > Porous Medium > Fluid > Lumped System Connector

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Lumped System Connector

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface selected:

Boundaries > interface > Lumped System Connector

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Lumped System Connector

Opaque Surface (Surface-to-Surface Radiation Interface)

This node is a variant of the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node with specular reflectivity. It is available when the **Surface-to-surface radiation method** is set to **Ray shooting** in the settings of the Surface-to-Surface Radiation interface. It applies to glossy surfaces where specular radiation should be considered in addition to diffuse radiation. The node adds to its selection a radiosity shape function for each spectral band, and uses it as surface radiosity.

If only diffuse reflection is to be taken into account, use the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node instead.

If pure specular reflection should be considered, use the [Specular Mirror \(Surface-to-Surface Radiation Interface\)](#) node instead.

It is assumed that no radiation is transmitted through the surface. If radiative transmission is also to be taken into account, use the [Semitransparent Surface \(Surface-to-Surface Radiation Interface\)](#) node instead.

MODEL INPUT

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups have been added, the model inputs are included here.

There is one standard model input: the **Temperature** T is used in the expression of the blackbody radiation intensity and when multiple wavelength intervals are used, for the fractional emissive power. The temperature model input is also used to determine the variable that receives the radiative heat source. When the model input does not contain a dependent variable, the radiative heat source is ignored.

The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

COORDINATE SYSTEM SELECTION

Select a boundary system from the **Coordinate system** list to define the reference vector for the azimuthal angle. The first component of the boundary system selected is the reference vector used for directional surface properties. The default is the boundary system of the model, and the list contains any additional boundary system added under the **Definitions** node.

See [Coordinate Systems](#) in the *COMSOL Multiphysics Reference Manual* for more details.

RADIATION DIRECTION

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

AMBIENT

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

FRACTIONAL EMISSIVE POWER

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

SURFACE RADIATIVE PROPERTIES

Directional Dependence

By default, the **Directional dependence** of the surface emissivity is set to **Constant**. The **Emissivity** can be defined by an expression depending on other variables such as temperature or spatial coordinates. Switch to the **Polar** or **Polar and azimuthal** options to define the dependencies on the polar and azimuthal angles (θ and φ , respectively) 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 $f_\epsilon(\theta) = 0$ or $f_\epsilon(\theta, \varphi) = 0$.

The other dependencies (temperature T , spatial coordinates $\mathbf{x}$) are treated separately in the **Emissivity** expression, and used to define the total emissivity:

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

Select the **Define properties on each side** checkbox to define specific directional dependencies of the surface emissivity on each side of the boundary, and set the **Directional dependence, upside** and **Directional dependence, downside** accordingly.



The reference vector for the azimuthal angle is given by the first component of the **Coordinate system** selected in the **Coordinate System Selection** section.

Emissivity, Directional Emissivity Function, and Diffuse Reflectivity

If the **Wavelength dependence of radiative properties** is **Constant**:

- By default, **Emissivity** ϵ (dimensionless) and **Diffuse reflectivity** ρ_d (dimensionless) use values **From material**. These are properties of the surface that depend both on the material itself and the structure of the surface. Make sure that a material is defined at the boundary level (by default materials are defined at the domain level).
- For **User defined**, set values or expressions. You can define the emissivity and reflectivity as functions of temperature using the variable `rad.T`.

Select the **Define properties on each side** checkbox to define specific values for each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside**, **Emissivity, downside**, **Diffuse reflectivity, upside**, and **Diffuse reflectivity, downside** are **From material**.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Emissivity** ε (dimensionless) and **Diffuse reflectivity** ρ_d (dimensionless) use values **From material**.
- When **Emissivity** is **User defined**, enter a value or expression for **Emissivity** ε . The wavelength is accessible through the `rad.lambda` variable. Any expression set for the emissivity is then averaged over each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.
- When **Diffuse reflectivity** is **User defined**, enter a value or expression for the **Diffuse reflectivity** ρ_d . The wavelength is accessible through the `rad.lambda` variable. Any expression set for the reflectivity is then averaged on each spectral band to obtain a piecewise constant reflectivity. If the average value of the reflectivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.
- When **Emissivity** is **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is set on both sides. Select the **Define properties on each side** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.
- When **Diffuse reflectivity** is **User defined for each band**, enter a value for the **Diffuse reflectivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same reflectivity is defined on both sides. Select the **Define properties on each side** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

Set the surface emissivity to a number between 0 and 1, where 0 represents a diffuse mirror and 1 is suitable for a perfect blackbody. The appropriate value for a physical material is somewhere in between and can be found in tables or measurements.

Set the diffuse reflectivity of the surface to a number between 0 and 1, where 1 applies to a perfect diffuse surface and 0 is appropriate when the reflection is only specular.

The specular surface reflectivity, ρ_s , is then defined as

$$\rho_s = 1 - \rho_d - \varepsilon$$

When the **Emitted radiation direction** is set to **None** for a spectral band, the information defined for that spectral band in this section is not used.

LOCATION IN USER INTERFACE

Context Menus

Surface-to-Surface Radiation > Opaque Surface

Ribbon

Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries > Surface-to-Surface Radiation > Opaque Surface

Opaque Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node defines a boundary that is opaque to radiation. It prescribes the incident intensities on a boundary and takes into account the net radiative heat flux, $q_{r,net}$, which is absorbed by the surface.

The net radiative heat flux is defined as the difference between the incoming and outgoing radiative heat fluxes:

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

The incoming and outgoing radiative heat fluxes are defined from the weighted sums of the incident intensities.

See [Opaque Surface](#) for more details on the definition of the radiative intensities on the boundary.

If the boundary is semitransparent, consider using the [Semitransparent Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) node instead.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must be created first. See [Identity](#), [Contact](#), and [Sector Symmetry 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** checkbox to set specific **Upside** and **Downside** values in the table.

SURFACE RADIATIVE PROPERTIES

Select a **Surface type** to define the behavior of the surface: **Gray surface** or **Black surface**. The definition of the incident intensities used to define the incoming and outgoing radiative heat fluxes depends on the option selected in this list.

Gray Surface

If **Gray surface** is selected, the surface emissivity should be set. This is a property of the material surface that depends both on the material itself and the structure of the surface.

An emissivity of 0 means that the surface does not emit any radiation and that all outgoing radiation is diffusely reflected by this boundary. An emissivity of 1 means that the surface is a perfect blackbody: all outgoing radiation is fully absorbed on this boundary. When the node is added in the Radiation in Participating Media interface

with the Discrete Ordinates Method, the radiative intensity along the discrete incoming directions on this boundary is defined by

$$I_i = \varepsilon I_b(T) + \frac{1-\varepsilon}{\pi} q_{r, out}$$

When it is added in the Radiation in Absorbing-Scattering Media interface with the Discrete Ordinates Method, no emission is taken into account, and the radiative intensity along the incoming discrete directions on this boundary is defined by

$$I_i = \frac{1-\varepsilon}{\pi} q_{r, out}$$

The default value of the **Emissivity** ε is taken **From material**. Make sure a material is defined at the boundary level (by default materials are defined at the domain level).

For **User defined** enter another value or expression. You can define a temperature-dependent emissivity using the variable `rpm.T` or `rasm.T`.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, the wavelength can be accessed by the `rpm.lambda` or `rasm.lambda` variable. Any expression defined for the emissivity is then averaged over each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you may use instead the option **User defined for each band** to avoid the evaluation of the average.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, alternatively set the **Emissivity** to **User defined for each band** to enter a value for each spectral band in the table shown below. Within a spectral band, each value is assumed to be independent of wavelength.

Select the **Define properties on each side** checkbox to set specific values on each side of the boundary. The **Emissivity, upside** and **Emissivity, downside** should be set, with the same options as those described above.

Black Surface

If **Black surface** is selected, no user input is required, and the radiative intensity along the discrete incoming directions on this boundary is defined by

$$I_i = I_b(T)$$

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
	<i>Radiative Heat Transfer in a Utility Boiler</i> : Application Library path Heat_Transfer_Module/Thermal_Radiation/boiler

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.



Radiative Beam in Absorbing Media Theory

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

Open Boundary

Use this condition to define a boundary as the limit between a nonsolid 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:

$$-\mathbf{n} \cdot \mathbf{q} = 0, \quad \mathbf{n} \cdot \mathbf{u} \geq 0 \quad (6-17)$$

For an incoming flow of velocity field $\mathbf{u}$ across the boundary, it accounts by default for the heat flux induced by the flow rate through a Danckwerts condition on the enthalpy:

$$-\mathbf{n} \cdot \mathbf{q} = \rho \Delta H \mathbf{u} \cdot \mathbf{n}, \quad \mathbf{n} \cdot \mathbf{u} < 0 \quad (6-18)$$

where ΔH is the sensible enthalpy. See [Theory for the Inflow Boundary Condition](#) for more details on the Danckwerts condition and the definition of ΔH .

Alternatively the open boundary condition can set a constraint on the temperature for an incoming flow

$$T = T_{\text{ustr}}, \quad \mathbf{n} \cdot \mathbf{u} < 0 \quad (6-19)$$

Both conditions are based on the knowledge of the upstream temperature at the outer boundary of the virtual domain, but the first one should be preferred to take into account the feedback of the model heat sources and temperature constraints on the inlet temperature profile.

UPSTREAM PROPERTIES

Enter the **Upstream temperature** T_{ustr} outside the open boundary. For **User Defined**, enter a value or expression. Else, select any available input (like an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**).

INFLOW CONDITION

Unfold this section to change the inflow condition:

- The **Flux (Danckwerts)** condition (default) prescribes the flux defined from the upstream temperature and the fluid velocity at the boundary. Using this condition the temperature condition is not strictly enforced at the inlet. Instead a flux is set on this boundary. The higher the flow rate, the smallest difference between the inlet temperature and upstream temperature.
- The **Nitsche constraints** option enforces the upstream temperature on the boundary..



Natural Convection Cooling of a Vacuum Flask: Application Library
path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/
vacuum_flask**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Flow Conditions > Open Boundary
Heat Transfer in Porous Media > Porous Medium > Fluid > Flow Conditions > Open Boundary
Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Flow Conditions > Open Boundary

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries > interface > Open Boundary

Physics tab with **Porous Medium > Fluid** selected in the model tree:

Attributes > Open Boundary

Outflow

This node provides a suitable boundary condition for convection-dominated heat transfer at outlet boundaries. In a model with convective heat transfer, this condition states that the only heat transfer occurring across the boundary is by convection. The temperature gradient in the normal direction is zero, and there is no radiation. This is usually a good approximation of the conditions at an outlet boundary in a heat transfer model with fluid flow.

BOUNDARY SELECTION

The **Outflow** node does not require any user input. If required, select the boundaries that are convection-dominated outlet boundaries.



Heat Sink: Application Library path **Heat_Transfer_Module/**
Tutorials,_Forced_and_Natural_Convection/heat_sink

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Flow Conditions > Outflow

Heat Transfer in Porous Media > Porous Medium > Fluid > Flow Conditions > Outflow

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Flow Conditions > Outflow

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, or any version of the Heat Transfer interface selected:

Boundaries > interface > Outflow

Physics tab with **Porous Medium > Fluid** selected in the model tree:

Attributes > Outflow

Periodic Condition (Heat Transfer Interface)

Use this node to add periodic temperature conditions to boundary pairs. The **Destination Selection** section is available if you select the **Advanced Physics Options** in the **Show More Options** dialog.

For information about the **Orientation of Source** and **Orientation of Destination** sections, 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.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Nitsche constraints** (default), **Pointwise constraints** or **Weak constraints**.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	<i>Convection Cooling of Circuit Boards — 3D Forced Convection:</i> 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*.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings

LOCATION IN USER INTERFACE

Context Menus

Radiation in Participating Media > Periodic Condition
Radiation in Absorbing-Scattering Media > Periodic Condition

Ribbon

Physics tab with **Radiation in Participating Media** or **Radiation in Absorbing-Scattering Media** selected:

Boundaries > Periodic Condition

Phase Change Interface

Use this node to define the temperature and the position of the interface between two phases of the same material, using the Stefan condition. It is applicable on boundaries

between [Fluid](#) and [Solid](#) domains, and on interior boundaries of [Fluid](#), [Solid](#), and [Porous Medium](#) domains, in 2D, 2D axisymmetric, and 3D components.

For an equivalent condition applicable on exterior boundaries, use [Phase Change Interface](#), [Exterior](#) instead.

The following hypotheses are made when using this node:

- One of the phases, the one that corresponds to the coldest phase, should be immobile. The properties of this phase are denoted by subscript s, which stands for solid, but this phase can be an immobile fluid as well. The properties of the other phase are denoted by subscript f, which stands for fluid.
- In a porous medium, the porous matrix is not subject to phase change, which occurs only in the fluid material. The effective density of the phase change material used in the Stefan condition depends on the porosity. Thus, a [Porous Medium](#) node should be active on each side of the phase change interface.
- Mass conservation should be verified. This can be ensured by using a velocity field calculated by a Fluid Flow interface in the fluid domain. If [Solid](#) nodes are used for both phases, they should have the same density, to ensure mass conservation on the material frame.
- No topology changes are allowed for the interface. If some topology change is expected, the [Phase Change Material](#) subnode should be used instead.

In stationary and time-dependent studies, the temperature is set to the phase change temperature on the interface:

$$T = T_{pc}$$

In addition, in time-dependent studies, the Stefan condition defines the phase change interface velocity v_n from the conductive heat flux jump across the interface, $\mathbf{q}$, the latent heat of phase change from solid to fluid, $L_{s \rightarrow f}$, and the solid density, ρ_s :

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

with

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

The fluid velocity at the interface, v_f , is defined as:

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

The phase change velocity v_n appearing in Stefan condition is relative to the solid position. In case of translation of the solid (using [Translational Motion](#) subfeature under [Solid with Translational Motion](#) feature for instance), the solid translation velocity $\mathbf{u}_s$ contributes to v_n to describe the interface velocity relative to the spatial frame:

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

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

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

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**.



The red arrow indicates the location of the solid phase, relatively to the selected boundaries in the **Graphics** window. Upside and downside settings are based on the global normal vector (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*.

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 degree of accuracy as 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** checkbox 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 tradeoff between numerical stability and precision of the physical modeling.

	See Arbitrary Lagrangian-Eulerian Method .
	<i>Tin Melting Front</i> : Application Library path Heat_Transfer_Module/Phase_Change/tin_melting_front <i>Continuous Casting — Arbitrary Lagrangian-Eulerian Method</i> : Application Library path Heat_Transfer_Module/Thermal_Processing/continuous_casting_ale

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Phase Change Interface

Heat Transfer in Solids > Phase Change Interface

Heat Transfer in Fluids > Phase Change Interface

Heat Transfer in Porous Media > Phase Change Interface

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, or **Heat Transfer in Porous Media** selected:

Boundaries > interface > Phase Change Interface

Phase Change Interface, Exterior

Use this node to define the temperature and the position of the interface of a phase change material, using the Stefan condition. It is applicable on exterior boundaries adjacent to [Fluid](#), [Solid](#), and [Porous Medium](#) domains, in 2D, 2D axisymmetric, and 3D components.

For an equivalent condition on interior boundaries, use the [Phase Change Interface](#) node instead. The settings for these two nodes are similar, except that some properties entering in the expression of the Stefan velocity should be specified on exterior

boundaries, depending on which side of the phase change interface the solid phase is located, outside or inside the computational domain. See the [Phase Change Interface](#) and [Heat Flux](#) sections for details. On interior boundaries the heat flux and solid phase properties are computed from the adjacent domains.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat flux q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See [Conversion Between Material and Spatial Frames](#) for details.
- The default option for the **Phase Change Interface, Exterior** node is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the boundary on which the node is active.



This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See [Handling Frames in Heat Transfer](#) and [Material and Spatial Frames](#) for details.

PHASE CHANGE INTERFACE

This section defines the parameters used in the Stefan condition.

First, set the **Phase change temperature**, T_{pc} (SI unit: K), and the **Latent heat**, $L_{s \rightarrow f}$ (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 **Inside** or **Outside** the computational domain.

When the solid phase is outside the computational domain, the **Solid density**, ρ_s (SI unit: kg/m^3), and the **Solid translation velocity**, $\mathbf{u}_s$ (SI unit: m/s), should be specified.

When the solid phase is inside the computational domain, the **Fluid density**, ρ_f (SI unit: kg/m^3) should be specified.

Both solid and fluid densities are used to evaluate the fluid velocity at the interface, v_f , defined as:

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



The red arrow indicates the location of the solid phase, relatively to the selected boundaries in the **Graphics** window.

Finally, choose the method of **Heat flux jump evaluation**. By default the **Lagrange multiplier** option is selected and introduces a variable on the boundary, the Lagrange multiplier, to prescribe the temperature and calculate the heat flux jump accurately.

Alternatively, select **Temperature gradient** to prescribe the temperature condition using a strong constraint and evaluate the heat flux jump from the temperature field gradient at the interface.

The **Temperature gradient** option leads to a formulation that is easier to handle from a numerical point of view, especially for the iterative solvers, however the accuracy of the heat flux jump is strongly dependent on the size of the mesh next to the boundary. In this case a very fine mesh may be needed to reach the same degree of accuracy as the default option with a default mesh. See [Weak Constraints](#) in the *COMSOL Multiphysics Reference Manual* for more details on the use of Lagrange multipliers.

MOVING BOUNDARY SMOOTHING

See [Moving Boundary Smoothing](#) in the **Phase Change Interface** node for details about this section.

HEAT FLUX

Click to select the **General inward heat flux** (default), **Convective heat flux**, or **Heat rate** button. Note that the **Convective heat flux** option is only available when the solid phase is inside the computational domain. The settings are the same as in the **Heat Flux** node, see [Heat Flux](#) for details.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Phase Change Interface, Exterior

Heat Transfer in Solids > Phase Change Interface, Exterior

Heat Transfer in Fluids > Phase Change Interface, Exterior

Heat Transfer in Porous Media > Phase Change Interface, Exterior

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, or **Heat Transfer in Porous Media** selected:

Boundaries > interface > Phase Change Interface, Exterior

Prescribed Radiosity (Surface-to-Surface Radiation Interface)

Use this node to specify radiosity on the boundary for each spectral band. Radiosity can be defined as *blackbody* or *graybody radiation*. A user-defined surface radiosity expression can also be defined.

COORDINATE SYSTEM SELECTION

Select a boundary system from the **Coordinate system** list to define the reference vector for the azimuthal angle. The first component of the boundary system selected is the reference vector used for directional surface properties. The default is the boundary system of the model, and the list contains any additional boundary system added under the **Definitions** node.

Directional dependence of radiative properties is available when the **Surface-to-surface radiation method** is set to **Ray shooting** in the settings of the Surface-to-Surface Radiation interface.

See [Coordinate Systems](#) in the *COMSOL Multiphysics Reference Manual* for more details.

RADIATION DIRECTION

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

RADIOSITY



Radiosity does not directly affect the boundary condition on the boundary where it is specified, but rather defines how that boundary affects others through radiation.

If **Wavelength dependence of radiative properties** is **Constant**, select a **Radiosity expression** between **Graybody radiation** (default), **Blackbody radiation**, or **User defined**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**, select a **Radiosity expression** between **Graybody radiation** (default), **Blackbody radiation**, **User defined for each band**, or **User defined**.

Blackbody Radiation

When **Blackbody radiation** is selected it defines the surface radiosity expression corresponding to a blackbody. Define the **External temperature** T_{ext} (SI unit: K, default value 293.15 K) to specify the temperature at which the blackbody radiation intensity e_b is evaluated. Alternatively, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

- When **Wavelength dependence of radiative properties** is set to **Constant** it defines $J = e_b(T_{\text{ext}})$ when radiation is defined on one side or $J_u = e_b(T_{\text{ext}})$ and $J_d = e_b(T_{\text{ext}})$ when radiation is defined on both sides.
- When **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, it defines for each spectral band $J_i = \text{FEP}_i(T_{\text{ext}})e_b(T_{\text{ext}})$ when radiation is defined on one side or $J_{i,d} = \text{FEP}_{i,d}(T_{\text{ext}})e_b(T_{\text{ext}})$ and $J_u = \text{FEP}_{i,u}(T_{\text{ext}})e_b(T_{\text{ext}})$ when radiation is defined on both sides.



- The blackbody hemispherical total emissive power is defined by
$$e_b(T) = n^2 \sigma T^4$$

Graybody Radiation

When **Graybody radiation** is selected, it sets the surface radiosity expression corresponding to a graybody with a temperature T_{ext} .

Directional dependence of radiative properties is available when the **Surface-to-surface radiation method** is set to **Ray shooting** in the settings of the Surface-to-Surface Radiation interface. By default, the **Directional dependence** of the surface emissivity is set to **Constant**. The **Emissivity** can be defined by an expression depending on other

variables such as temperature or spatial coordinates. Switch to the **Polar** or **Polar and azimuthal** options to define the dependencies 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 $f_\epsilon(\theta) = 0$ or $f_\epsilon(\theta, \varphi) = 0$.

The other dependencies (temperature T , spatial coordinates $\mathbf{x}$) are treated separately in the **Emissivity** expression, and used to define the total emissivity:

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

Select the **Define properties on each side** checkbox to define specific directional dependencies of the surface emissivity on each side of the boundary, and set the **Directional dependence, upside** and **Directional dependence, downside** accordingly.



The reference vector for the azimuthal angle is given by the first component of the **Coordinate System** selected in the **Coordinate System Selection** section.

By default, the **Emissivity** ϵ is defined **From material**. In this case, make sure that a material is defined at the boundary level (materials are defined by default at the domain level). If **User defined** is selected for the **Emissivity**, enter another value for ϵ . For the **User defined for each band** option, set a value for each spectral band in the table. Within a spectral band, each value is assumed to be independent of wavelength.

If **Wavelength dependence of radiative properties** is set to **Constant**:

- When radiation is defined on one side, define the **Emissivity** ϵ and the **External temperature** T_{ext} to set $J = \epsilon e_b(T_{\text{ext}})$, or
- When radiation is defined on both sides, define the **Emissivity, upside** ϵ_u , **Emissivity, downside** ϵ_d and the **External temperature** T_{ext} . The surface radiosity on upside and downside is then defined by $J_u = \epsilon_u e_b(T_{\text{ext}})$ and $J_d = \epsilon_d e_b(T_{\text{ext}})$, respectively.

If **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, for all spectral bands:

- When radiation is defined on one side for the Bi spectral band, define the **Emissivity** ϵ_i and the **External temperature** T_{ext} to set $J_i = \text{FEP}_i \epsilon_i e_b(T_{\text{ext}})$, or

- When radiation is defined on both sides for Bi spectral band, define the **Emissivity, upside** ϵ_u , **Emissivity, downside** ϵ_d and the **External temperature** T_{ext} . The surface radiosity on upside and downside is then defined by $J_u = FEP_{Bi}(T_{ext})\epsilon_{Bi, u}e_b(T_{ext})$ and $J_d = FEP_{Bi}(T_{ext})\epsilon_{Bi, d}e_b(T_{ext})$, respectively.



Set the surface emissivity to a number between 0 and 1, where 0 represents diffuse mirror and 1 is appropriate for a perfect blackbody. The proper value for a physical material lies somewhere in between and can be found from tables or measurements.

User Defined

If **Wavelength dependence of radiative properties** is set to **Constant** and **Radiosity expression** is set to **User defined**, it sets the surface radiosity expression to $J = J_0$, which specifies how the radiosity of a boundary is evaluated when that boundary is visible in the calculation of the irradiation onto another boundary in the model. Enter a value or expression for the **Surface radiosity**, J_0 .

Select the **Define prescribed radiosity on each side** checkbox 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** checkbox 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 <i>COMSOL Multiphysics Reference Manual</i>.
	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** checkbox under **Radiation Settings** to add a **Radiation Group** to a Surface-to-Surface Radiation (rad) interface.

When a boundary is added to another radiation group, it is overridden in the default group. Then this boundary can be added to other radiation groups without being overridden by the manually added radiation groups.



Be careful when grouping boundaries in axisymmetric geometries. The grouping cannot be based on which boundaries have a free view toward each other in the 2D geometry. Instead, consider the full 3D geometry, obtained by revolving the model geometry about the z axis, when defining groups.

For example, parallel vertical boundaries must typically belong to the same group in 2D axisymmetric components, but to different groups in a planar model using the same 2D geometry.

The figure below shows four examples of possible boundary groupings. On boundaries that have no number, the user has NOT set a node among the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#), [Diffuse Mirror \(Surface-to-Surface Radiation Interface\)](#), and [Prescribed Radiosity \(Surface-to-Surface Radiation Interface\)](#) nodes. These boundaries do not irradiate other boundaries, neither do other boundaries irradiate them.

On boundaries that belong to one or more radiation group, the user has set a node among the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#), [Diffuse Mirror \(Surface-to-Surface Radiation Interface\)](#), and [Prescribed Radiosity \(Surface-to-Surface](#)

Radiation Interface) nodes. The numbers on each boundary specify different groups to which the boundary belongs.

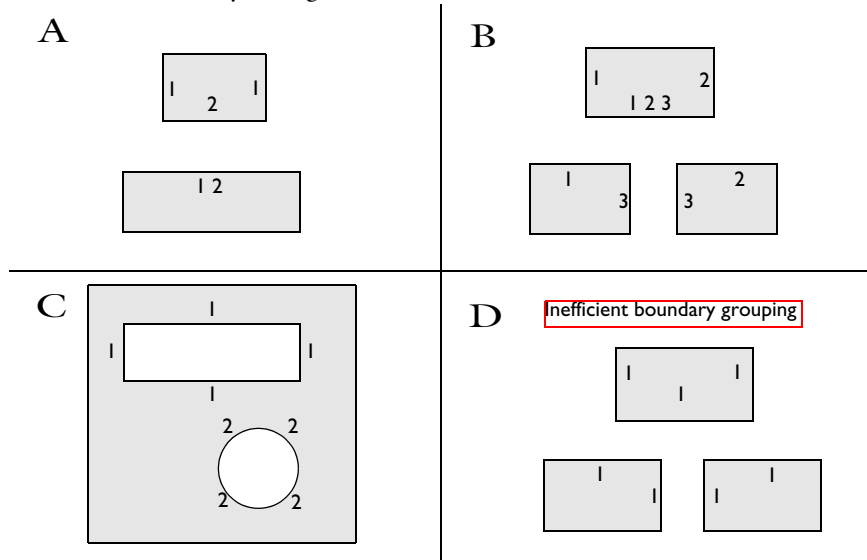


Figure 6-3: Examples of radiation group boundaries.

To obtain optimal computational performance, it is good practice to specify as many groups as possible as opposed to specifying few but large groups. For example, case B is more efficient than case D.

BOUNDARY SELECTION

This section should contain any boundary that is selected in a **Diffuse Surface**, **Diffuse Mirror**, or **Prescribed Radiosity** node and that has a chance to see one of the boundary that is already selected in the **Radiation Group**.

RADIATION GROUP

When the **Wavelength dependence of radiative properties** is **Constant**, the radiation group is valid for all wavelengths, and all this section is then empty.

When the **Wavelength dependence of radiative properties** is set to **Solar and ambient** or **Multiple spectral bands**, the radiation group is defined for all spectral bands by default.

Clear **Radiation group defined on spectral band i** checkboxes 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** checkbox is selected under the **Radiation Settings** section:

Surface-to-Surface Radiation > Radiation Group

Ribbon

Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries > Radiation Group

Semitransparent Surface (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node defines a boundary semitransparent to radiation. If no radiation is transmitted through the surface, use the [Opaque Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) node instead.

It prescribes incident intensities on a boundary and takes into account the net radiative heat flux absorbed by the surface, modeling emitted, reflected, and (diffusively and specularly) transmitted radiative intensity on the boundary.

When the **Radiation discretization method** is **Discrete ordinates method**, the net radiative heat flux is defined as the difference between the incoming and outgoing radiative heat fluxes, and the incoming and outgoing radiative heat fluxes are defined from the weighted sums of the incident intensities. See [Semitransparent Surface](#) for more details.

When the **Radiation discretization method** is **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, $q_{r,net,d}$ and $q_{r,net,u}$ on each side of the surface are defined. Specific radiative properties of the surface can be defined on each side of the boundary.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must be created first. See [Identity](#), [Contact](#), and [Sector Symmetry 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** checkbox to set specific **Upside** and **Downside** values in the table.

SURFACE RADIATIVE PROPERTIES

The radiation properties of the boundary and the external radiation intensity should be set in this section.

Emissivity, Diffuse Transmissivity, and Specular Transmissivity

If the **Wavelength dependence of radiative properties** is **Constant**:

- By default, the **Emissivity** ε (dimensionless), **Diffuse transmissivity** τ_d (dimensionless), and **Specular transmissivity** τ_s (dimensionless) use values **From material**. These are properties of the material surface that depend both on the material itself and the structure of the surface. Make sure a material is defined at the boundary level (by default materials are defined at the domain level).
- For **User defined**, set values or expressions. You can define temperature-dependent emissivity and transmissivities using the variable `rad.T`.

Select the **Define properties on each side** checkbox to set specific values for each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside**, **Emissivity, downside**, **Diffuse transmissivity, upside**, **Diffuse transmissivity, downside**, **Specular transmissivity, upside**, and **Specular transmissivity, downside** are **From material**.

If the **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, **Emissivity** ε (dimensionless), **Diffuse transmissivity** τ_d (dimensionless), and **Specular transmissivity** τ_s (dimensionless) use values **From material**.
- When **Emissivity** is **User defined**, enter a value or expression for the **Emissivity** ε . The wavelength is accessible through the `rad.lambda` variable. Any expression set for the emissivity is then averaged on each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.
- When **Diffuse transmissivity** is **User defined**, enter a value or expression for the **Diffuse transmissivity** τ_d . The wavelength is accessible through the `rad.lambda` variable. Any expression set for the transmissivity is then averaged on each spectral band to obtain a piecewise constant transmissivity. If the average value of the transmissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.
- When **Specular transmissivity** is **User defined**, enter a value or expression for the **Specular transmissivity** τ_s . The wavelength is accessible through the `rad.lambda`

variable. Any expression set for the specular transmissivity is then averaged on each spectral band to obtain a piecewise constant specular transmissivity. If the average value of the specular transmissivity on each band is known, you can use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Emissivity** is **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is defined on both sides. Select the **Define properties on each side** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.
- When **Diffuse transmissivity** is **User defined for each band**, enter a value for the **Diffuse transmissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same transmissivity is defined on both sides. Select the **Define properties on each side** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.
- When **Specular transmissivity** is **User defined for each band**, enter a value for the **Specular transmissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same transmissivity is defined on both sides. Select the **Define properties on each side** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

Set the surface emissivity to a number between 0 and 1, where 0 represents a diffuse mirror and 1 is suitable for a perfect blackbody. The appropriate value for a physical material is somewhere in between and can be found in tables or measurements.

Set the surface diffuse transmissivity to a number between 0 and 1, where 0 applies to an opaque surface and 1 is appropriate for a fully transparent surface.

Set the surface specular transmissivity to a number between 0 and 1, where 0 applies to a perfect diffuse surface and 1 is appropriate when transmissivity is only specular.



The **Specular transmissivity** is only needed when **Discrete ordinates method** is selected.

EXTERNAL RADIATION INTENSITY

The external radiation intensity should be defined when the boundary condition is applied to an external boundary. This intensity is a power per unit solid angle and unit surface area projected onto the plane normal to the direction of radiation. It can be specified for each spectral band.

When the **Wavelength dependence of radiative properties** is set to **Constant**, two options are available to define the external radiation intensity. Set **External radiation intensity** to **User defined** to define I_{ext} (SI unit: $\text{W}/\text{m}^2/\text{sr}$) directly. Alternatively, select **Blackbody radiation** to define I_{ext} as $I_b(T_{\text{ext}})$ and set the **External temperature** T_{ext} (SI unit: K, default value 293.15 K) or select an **Ambient temperature** to specify the temperature at which the blackbody radiation intensity I_b is evaluated.

When the **Wavelength dependence of radiative properties** is set to **Solar and ambient** or to **Multiple spectral bands**, the external radiation intensity, $I_{\text{ext},k}$ is defined for each spectral band.

Set **External radiation intensity** to **Blackbody radiation** and set the **External temperature** T_{ext} (SI unit: K, default value 293.15 K) or select an **Ambient temperature** to define $I_{\text{ext},k}$ from the blackbody intensity $I_b(T_{\text{ext}})$, the **External temperature** T_{ext} , and the fractional emissive power for each spectral band at external temperature $\text{FEP}_k(T_{\text{ext}})$.

Set the **External radiation intensity** to **User defined for each band**, to define $I_{\text{ext},k}$ from user defined expression for each spectral band in the table displayed underneath.

Set the **External radiation intensity** to **User defined distribution**, and define an expression for $I_{\lambda,\text{ext}}$ (SI unit: $\text{W}/\text{m}^3/\text{sr}$) to define $I_{\text{ext},k}$ from a spectral distribution. In this case $I_{\text{ext},k}$ is automatically computed per spectral band as the integral of the distribution over each spectral band. If this integral over each band is known, you can use instead the **User defined for each band** option to avoid the numerical overhead due to the evaluation of the average.



When **Discrete ordinates method** is selected, the components of each discrete ordinate vector can be used in this expression. The syntax is `name.sx`, `name.sy`, and `name.sz`, where `name` is the name of the physics interface node. By default, the Radiation in Participating Media interface is `rpm` so `rpm.sx`, `rpm.sy`, and `rpm.sz` correspond to the components of discrete ordinate vectors.



- [Participating Medium \(Radiation in Participating Media Interface\)](#)
- [Absorbing-Scattering Medium \(Radiation in Absorbing-Scattering Media Interface\)](#)
- [Theory for Radiation in Participating Media](#)

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 pure specular reflection should be considered, use the [Specular Mirror \(Surface-to-Surface Radiation Interface\)](#) node instead.

If no radiation is transmitted through the surface, use the [Opaque Surface \(Surface-to-Surface Radiation Interface\)](#) node instead.

Radiation is assumed to occur on both sides of the surface.

MODEL INPUT

This section has fields and values that are inputs to expressions that define material properties. If such user-defined property groups have been added, the model inputs are included here.

There is one standard model input: the **Temperature T** is used in the expression of the blackbody radiation intensity and when multiple wavelength intervals are used, for the fractional emissive power. The temperature model input is also used to determine the variable that receives the radiative heat source. When the model input does not contain a dependent variable, the radiative heat source is ignored.

The default **Temperature** is **User defined**. When additional physics interfaces are added to the model, the temperature variables defined by these physics interfaces can also be selected from the list. The **Common model input** option corresponds to the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

COORDINATE SYSTEM SELECTION

Select a boundary system from the **Coordinate system** list to define the reference vector for the azimuthal angle. The first component of the boundary system selected is the reference vector used for directional surface properties. The default is the boundary system of the model, and the list contains any additional boundary system added under the **Definitions** node.

See [Coordinate Systems](#) in the *COMSOL Multiphysics Reference Manual* for more details.

AMBIENT

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

FRACTIONAL EMISSIVE POWER

These settings are the same as for the [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#) node.

SURFACE RADIATIVE PROPERTIES

Directional Dependence

By default, the **Directional dependence** of the surface radiative properties (emissivity and transmissivity) is set to **Constant**. The **Emissivity** and **Transmissivity** can 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 dependencies 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 **Emissivity** and **Transmissivity** expressions, and used to define the total properties:

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

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

Emissivity, Diffuse Reflectivity, Transmissivity, and Critical Angle

If **Wavelength dependence of radiative properties** is **Constant**:

- By default, the **Emissivity** ε (dimensionless), **Diffuse reflectivity** ρ_d (dimensionless), and **Transmissivity** τ (dimensionless) use values **From material**. These are properties of the material surface that depend both on the material itself and the structure of the surface. Make sure that a material is defined at the boundary level (by default materials are defined at the domain level).
- For **User defined**, set values or expressions. You may set temperature-dependent emissivity and reflectivity using the variable `rad.T`.

Select the **Define properties on each side** checkbox to set specific values on each side. Select the **Boundary material, upside** and **Boundary material, downside** to have different material properties on each side. The boundary material specified is used only when **Emissivity, upside**, **Emissivity, downside**, **Diffuse reflectivity, upside**, **Diffuse reflectivity, downside**, **Transmissivity, upside**, **Transmissivity, downside** are **From material**.

If **Wavelength dependence of radiative properties** is **Solar and ambient** or **Multiple spectral bands**:

- By default, the **Emissivity** ε (dimensionless), **Diffuse reflectivity** ρ_d (dimensionless), and **Transmissivity** τ (dimensionless) use values **From material**.
- When **Emissivity** is set to **User defined**, enter a value or expression for the **Emissivity** ε . The wavelength may be accessed through the `rad.lambda` variable. Any expression set for the emissivity is then averaged on each spectral band to obtain a piecewise constant emissivity. If the average value of the emissivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.
- When **Diffuse reflectivity** is set to **User defined**, enter a value or expression for the **Diffuse reflectivity** ρ_d . The wavelength may be accessed through the `rad.lambda` variable. Any expression set for the reflectivity is then averaged on each spectral band to obtain a piecewise constant reflectivity. If the average value of the reflectivity on

each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.

- When **Transmissivity** is set to **User defined**, enter a value or expression for the **Transmissivity** τ . The wavelength may be accessed through the `rad.lambda` variable. Any expression set for the transmissivity is then averaged on each spectral band to obtain a piecewise constant transmissivity. If the average value of the transmissivity on each band is known, you may use instead the **User defined for each band** option to avoid the evaluation of the average.
- When **Emissivity** is set to **User defined for each band**, enter a value for the **Emissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same emissivity is defined on both sides. Select the **Define properties on each side** checkbox 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** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.
- When **Transmissivity** is set to **User defined for each band**, enter a value for the **Transmissivity** for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength. By default, the same transmissivity is defined on both sides. Select the **Define properties on each side** checkbox and fill the **Upside** and **Downside** columns of the table for a specific definition on each side.

Set the surface emissivity to a number between 0 and 1, where 0 represents a diffuse mirror and 1 is appropriate for a perfect blackbody. The proper value for a physical material lies somewhere in between and can be found from tables or measurements.

Set the surface diffuse reflectivity to a number between 0 and 1, where 1 applies to a perfect diffuse surface and 0 is appropriate when reflection is only specular.

Set the surface transmissivity to a number between 0 and 1, where 0 applies to an opaque surface and 1 is appropriate for a fully transparent surface.

The surface specular reflectivity, ρ_s , is then defined as

$$\rho_s = 1 - \rho_d - \varepsilon - \tau$$

Set a value or expression for the **Critical angle** θ_c , which is the minimum angle of incidence (measured between the ray and the normal to the surface) for transmission to occur. Below this angle, all the radiation is reflected. The $\pi/2$ default value ensures that total reflection does not occur by default.

When the **Radiating side** is set to **None** for a spectral band, the information set for this spectral band in this section is not used.

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

Heat Transfer in Porous Media > Porous Medium > Fluid > Heat Sources >

Surface-to-Ambient Radiation

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Heat Sources >

Surface-to-Ambient Radiation

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids** or any version of the Heat Transfer interface selected:

Boundaries > Surface-to-Ambient Radiation

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Surface-to-Ambient Radiation

Specular Mirror (Surface-to-Surface Radiation Interface)

This node is a variant of the [Opaque Surface \(Surface-to-Surface Radiation Interface\)](#) node with a surface emissivity and diffuse reflectivity equal to zero and the surface specular reflectivity, p_s , is then set to one. 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 which behave like ideal mirrors that reflect all irradiation specularly. Zero values of emissivity and diffuse reflectivity result in identically zero radiosity. It is then not needed to solve for this variable. The node replaces the radiosity shape function for each spectral band on its selection with the explicit value of zero. The radiative heat flux on a specular mirror boundary is zero.

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.

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 > Specular Mirror

Ribbon

Physics tab with **Surface-to-Surface Radiation** selected:

Boundaries > Surface-to-Surface Radiation > Specular Mirror

Symmetry (Heat Transfer Interface)

This node provides a boundary condition for symmetry boundaries. This boundary condition is similar to a [Thermal Insulation](#) condition, and it means that there is no heat flux across the boundary.



The symmetry condition only applies to the temperature field. It has no effect on the radiosity (surface-to-surface radiation) and on the radiative intensity (radiation in participating media). Use [Symmetry for Surface-to-Surface Radiation](#) to take into account the symmetry in the computation of the view factors.



Heat Generation in a Disc Brake: Application Library path
Heat_Transfer_Module/Thermal_Contact_and_Friction/brake_disc

LOCATION IN USER INTERFACE

Context Menus

- Heat Transfer in Solids and Fluids > Symmetry**
- Heat Transfer in Porous Media > Porous Medium > Fluid > Symmetry**
- Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Symmetry**

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface selected:

Boundaries > interface > Symmetry

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Symmetry

Symmetry (Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces)

This node provides a condition for symmetry boundaries. When the **Radiation discretization method** is **PI approximation**, it corresponds to a no radiative flux condition across the boundary.

When the **Radiation discretization method** is **Discrete ordinates method**, it constrains radiative intensities of symmetrical directions to be equal. In that case, the **Symmetry** feature is restricted to flat boundaries which normal is aligned with the main axes of the component:

- For 3D components, boundaries of normal aligned with the x -axis, y -axis, or z -axis can be selected.
- For 2D components, boundaries of normal aligned with the x -axis or y -axis can be selected.
- For 2D axisymmetric components, only boundaries of normal aligned with the z -axis can be selected.



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 > 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 be created first. See [Identity](#), [Contact](#), and [Sector Symmetry 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** in the **Show More Options** dialog. The **Constraint** settings can be set to **Pointwise constraints** (default), **Weak constraints** or **Nitsche constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation. Select **Nitsche constraints** when **Pointwise constraints** and **Weak constraints** do not work satisfactorily.



The **Nitsche constraints** option is especially useful to prevent oscillations on inlet boundaries where convection dominates. Unlike the **Pointwise constraints** and **Weak 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 **Nitsche constraints** are not supported for resistive thin layers or with turbulent wall functions.



- [Constraint Reaction Terms](#)
- [Weak Constraints](#)
- [Constraint Settings](#)



You can specify a harmonic variation of the temperature value in a frequency domain analysis of perturbation type, by adding a **Harmonic Perturbation** subnode. See [Theory for Harmonic Heat Transfer](#) for more details.



Steady-State 2D Axisymmetric Heat Transfer with Conduction:
Application Library path **Heat_Transfer_Module/Tutorials,_Conduction/
cylinder_conduction**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Temperature

Heat Transfer in Porous Media > Porous Medium > Fluid > Temperature

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Temperature

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface selected:

Boundaries > interface > Temperature

Pairs > interface > Temperature

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Temperature

Thermal Contact

This node defines correlations for the conductance h at the interface of two bodies in contact. It can be added to internal boundaries by selecting **Thermal Contact** from the **Thin Structure** menu or to pairs by selecting **Thermal Contact** from the **Pairs** menu. Note that in this case the source term is applied on the destination side.

The conductance h is involved in the heat flux across the surfaces in contact according to:

$$-\mathbf{n}_d \cdot \mathbf{q}_d = -h(T_u - T_d) + rQ_b$$

$$-\mathbf{n}_u \cdot \mathbf{q}_u = -h(T_d - T_u) + (1 - r)Q_b$$

where u and d subscripts refer to the upside and downside of the slit, respectively. Pair version of **Thermal Contact** should be activated on a **Identity Pair** or on a **Contact Pair** where a structural mechanics physics interface defines a contact pair feature.

PAIR SELECTION

If this node is selected from the **Pairs** menu, choose the pair to apply this condition to. A pair must be created first. See [Identity, Contact, and Sector Symmetry 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} , the **Layer thickness**, d_s , the **Total resistance**, R_{tot} , and the **Total conductance**, h_{tot} . The last two options define the resistive behavior of the equivalent layer independently of the contact area.

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 <i>COMSOL Multiphysics Reference Manual</i>.



Thermal Contact Resistance Between an Electronic Package and a Heat Sink: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/thermal_contact_electronic_package_heat_sink**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Thin Structures > Thermal Contact

Heat Transfer in Solids > Thin Structures > Thermal Contact

Heat Transfer in Fluids > Thin Structures > Thermal Contact

Heat Transfer in Porous Media > Thin Structures > Thermal Contact

Bioheat Transfer > Thin Structures > Thermal Contact

Heat Transfer in Solids > Pairs > Thermal Contact

Heat Transfer in Fluids > Pairs > Thermal Contact

Heat Transfer in Porous Media > Pairs > Thermal Contact

Bioheat Transfer > Pairs > Thermal Contact

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* > Thermal Contact

Thermal Insulation

This node is the default boundary condition for all Heat Transfer interfaces. This boundary condition means that there is no heat flux across the boundary:

$$-\mathbf{n} \cdot \mathbf{q} = 0$$

and hence specifies where the domain is well insulated. Intuitively, this equation says that the temperature gradient across the boundary is zero. For this to be true, the temperature on one side of the boundary must equal the temperature on the other side. Because there is no temperature difference across the boundary, heat cannot transfer across it. As the default boundary condition, it can be applied only on external boundaries; but when added manually, it can be applied also on interior boundaries.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Thermal Insulation

Heat Transfer in Porous Media > Porous Medium > Fluid > Thermal Insulation

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > Thermal Insulation

Ribbon

Physics tab with *interface* as **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface selected:

Boundaries > *interface* > Thermal Insulation

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Thermal Insulation

Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface)

This node behaves like [Fluid](#) but is applicable on boundaries.

BOUNDARY SELECTION

For a manually added **Fluid** node in the Heat Transfer in Films interface, or for a **Thin Film** node in the Heat Transfer interface, select the **Restrict to layered boundaries** checkbox to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

The default **Fluid** node of the Heat Transfer in Films interface 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** checkbox 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** checkbox 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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes for the selection of the individual layers.
- Alternatively, set **Shell type** to **Nonlayered shell**, and set a user-defined value or expression for the **Thickness** L_{th} . This option should be used for thermally thin layers, for which no through-thickness temperature variation is expected in the layered material. This makes **Thermally thin approximation** the only option available for **Layer type** in the **Layer Model** section.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

Absolute Pressure

Absolute pressure is used in certain predefined quantities that include enthalpy (the energy flux, for example).

It is also used if the ideal gas law is applied. See [Thermodynamics, Fluid](#).

The default **Absolute pressure** p_A is **User defined**. When additional physics interfaces are added to the model, the absolute pressure variables defined by these physics interfaces

can also be selected from the list. For example, if a **Laminar Flow** interface is added you can select **Absolute pressure (spf)** from the list. The **Common model input** option corresponds to the `minput.pA` variable, set to 1 atm by default. To edit it, click the **Go to Source** button () , and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Pressure** in the **Expression for remaining selection** section.

LAYER MODEL

The available options for **Layer type** are **Thermally thin approximation** and **General** (when the **Shell type** is **Layered material** in the [Shell Properties](#) section). The former is a lumped model that accounts only for tangential temperature gradients (along the film), whereas the latter accounts also for the normal gradients of temperature (through the film's thickness). The second model may be used for the modeling of bearings for example.

If the **Layer type** is **Thermally thin approximation**, and the **Shell type** is **Layered shell**, the **Layerwise constant properties** checkbox is available to solve the equations with weighted averaged material properties, assuming that these properties are constant within each layer. When suitable, this allows to compute efficiently the homogenized properties of the layered material. When this checkbox is not selected, the properties are integrated through the thickness of the layered material. It means that you can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate, as described in [Using the Extra Dimension Coordinates](#).

HEAT CONVECTION

The settings are the same as in the [Fluid](#) domain feature.

HEAT CONDUCTION, FLUID

The settings are the same as in the [Fluid](#) domain feature.

THERMODYNAMICS, FLUID

The settings are the same as in the [Fluid](#) domain feature, except for the available options in the **Fluid type** list: **Ideal gas**, **Gas/Liquid**, and **Moist air**.

See the [Fluid](#) domain feature for more details on the **Ideal gas** and **Gas/Liquid** options.

Moist Air

If **Moist air** is selected, the thermodynamics properties are defined as a function of the amount of vapor in moist air. The **Input quantity** options available to define this amount are as follows:

- **Vapor mass fraction** to define the ratio between vapor mass and total mass. Enter a value or expression for the **Vapor mass fraction** ω .
- **Concentration** to define the amount of water vapor in the total volume. If selected, a **Concentration** model input is added to the **Model Inputs** section.
- **Moisture content** (default), also called mixing ratio or humidity ratio, to define the ratio between water vapor mass and dry air mass. For **User defined**, enter a value or expression for the **Moisture Content** x_{vap} . Else, select an **Ambient moisture content** defined in an **Ambient Properties** node under **Definitions**.
- **Relative humidity** ϕ_w , a quantity defined between 0 and 1, where 0 corresponds to dry air and 1 to air saturated with water vapor. The **Relative humidity, temperature condition** T_{ϕ_w} and **Relative humidity, absolute pressure condition** p_{ϕ_w} should be specified.



Within a layered material selection, a single **Layer type** should be used. If two layer types are needed for the same layered material, the original material should be duplicated so that one layered material is defined for each layer type. A [Shell Continuity \(Heat Transfer Interface\)](#) and [Continuity \(Heat Transfer in Shells Interface\)](#) node may be added between the two layered materials.



The thermal conductivity of the thin film is assumed to be isotropic, even if an anisotropic property is defined for the material.

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 sublayers with specific thickness and thermal properties. Each sublayer 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 sublayers 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 sublayers with specific thickness and thermal properties, and heat sources can be applied on a sublayer selection, and on up and down sides of the layer.

BOUNDARY SELECTION

For a manually added **Solid** node in the Heat Transfer in Shells interface, or for a **Thin Layer** node in the Heat Transfer interface, select the **Restrict to layered boundaries** checkbox to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

The default **Solid** node of the Heat Transfer in Shells interface 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** checkbox 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** checkbox 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** checkbox, when the node was added manually. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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** checkbox.

For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes for the selection of the individual layers.

- Alternatively, set **Shell type** to **Nonlayered shell**, and set a user-defined value or expression for the **Thickness** L_{th} . This option should be used for thermally thin layers, for which no through-thickness temperature variation is expected in the layered material. This removes the **General** option from the **Layer type** list in the **Layer Model** section.

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.



This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

LAYER MODEL

The available options for **Layer type** are **Thermally thin approximation**, **Thermally thick approximation** (available only in the **Thin Layer** feature), and **General**.

If **Layer type** is **Thermally thick approximation**, from the **Specify** list select **Layer properties** (default) or **Thermal resistance**.

If the **Layer type** is **Thermally thin approximation**, and the **Shell type** is **Layered shell**, the **Layerwise constant properties** checkbox is available to solve the equations with weighted averaged material properties, assuming that these properties are constant within each layer. When suitable, this allows to compute efficiently the homogenized properties of the layered material. By default this checkbox is not selected, and the properties are integrated through the thickness of the layered material. It means that you can provide material parameters with a through-thickness variation by using

expressions containing the extra dimension coordinate, as described in [Using the Extra Dimension Coordinates](#).



Within a layered material selection, a single **Layer type** should be used. If two layer types are needed for the same layered material, the original material should be duplicated so that one layered material is defined for each layer type. A [Shell Continuity \(Heat Transfer Interface\)](#) and [Continuity \(Heat Transfer in Shells Interface\)](#) node may be added between the two layered materials.

HEAT CONDUCTION

The default **Thermal conductivity** k is taken **From layered material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** to enter another value or expression.

Alternatively, set a value for the **Thermal resistance** R_s if **Specify** is set to **Thermal resistance** in the **Layer Model** section (for **Thermally thick approximation**).

THERMODYNAMICS

By default the **Density** ρ and **Heat capacity at constant pressure** C_p of the layer are taken **From layered material**. See [Material Density in Features Defined in the Material Frame](#) if a temperature-dependent density should be set. For **User defined** enter other values or expressions.

When **Layer type** is **Thermally thick approximation**, these properties are only used in time-dependent studies, but must be set in all cases.



These subnodes are available for **Thin Layer** node:

- [Heat Source \(Thin Layer, Thin Film, Fracture\)](#) — to add a layer internal heat source, Q_s , within the layer.
- [Heat Flux \(Thin Layer, Thin Film, Fracture\)](#) — to add a heat flux through a specified set of boundaries.
- [Temperature \(Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface\)](#) — to set a prescribed temperature condition on a specified set of boundaries.
- [Surface-to-Ambient Radiation \(Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface\)](#) — to add a surface-to-ambient radiation for the layer end.

	When multiple layers are defined they are numbered from the downside (Layer 1) to the upside. Upside and downside settings can be visualized by plotting the global normal vector (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 <i>COMSOL Multiphysics Reference Manual</i>.
	• Theory for Heat Transfer in Thin Structures • Boundary Wall Temperature • Plotting and Evaluating Results in Layered Materials
	• <i>Heat Transfer in a Surface-Mount Package for a Silicon Chip:</i> Application Library path Heat_Transfer_Module/Power_Electronics_and_Electronic_Cooling/surface_mount_package • <i>Silica Glass Block Coated with a Copper Layer:</i> 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 in Shells interfaces have the following boundary interface nodes and subnodes available:

- Deposited Beam Power, Interface (Heat Transfer in Shells Interface)
- Heat Flux, Interface (Heat Transfer in Shells Interface)
- Heat Source, Interface (Heat Transfer in Shells Interface)
- Lumped System Connector, Interface (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)
- Temperature, Interface (Heat Transfer in Shells Interface)
- Thermal Contact, Interface (Heat Transfer in Shells Interface)
- Thermal Insulation, Interface (Heat Transfer in Shells Interface)



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

Deposited Beam Power, Interface (Heat Transfer in Shells Interface)

This node models heat sources brought by narrow beams, such as laser or electron beams, to the exterior interfaces of a shell and, for the case of a layered shell, to the interfaces between its layers.

BOUNDARY SELECTION

Select the boundaries on which to apply the heat source.

In addition, select the **Restrict to layered boundaries** checkbox 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** checkbox 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 checkboxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See [Interface Selections](#) for more details on the interfaces identification.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**. This setting has no effect unless the temperature differs from one side of the boundary to the other.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.



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.

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 **Beam power density**, I_0 (SI unit: W/m^2). The **Beam power density** corresponds to the beam intensity in the beam cross section.

For **Built-in beam profiles**, enter a value for the **Deposited beam power** P_0 and the coordinates of the **Beam origin point** $\mathbf{O}$.

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** checkbox 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** checkbox 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 checkboxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See [Interface Selections](#) for more details on the interfaces identification.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**. This

setting has no effect unless the temperature differs from one side of the boundary to the other.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	- For a general description of layer and interface selections, see Layer and Interface Selection Tools. - You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.
	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 <i>COMSOL Multiphysics Reference Manual</i>.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat fluxes q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See [Conversion Between Material and Spatial Frames](#) for details.

- The default option for the **Heat Flux** node is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.

	This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See Handling Frames in Heat Transfer and Material and Spatial Frames for details.

HEAT FLUX

These settings are the same as for the [Heat Flux](#) node.

	In 2D the heat flux contribution is multiplied by d_z to take into account the out-of-plane thickness.

	• Theory for Heat Transfer in Thin Structures • Convective Heat Transfer Correlations

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Interfaces > Heat Flux, Interface

Heat Transfer in Films > Interfaces > Heat Flux, Interface

Heat Transfer in Fractures > Interfaces > Heat Flux, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries > Heat Flux, Interface

Heat Source, Interface (Heat Transfer in Shells Interface)

This node models a heat source (or sink) on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers. It adds a heat source $q = Q_b$ or $q = P_b/A$. A positive q is heating and a negative q is cooling.

BOUNDARY SELECTION

Select the boundaries on which to apply the heat source.

In addition, select the **Restrict to layered boundaries** checkbox 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** checkbox 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 checkboxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See [Interface Selections](#) for more details on the interfaces identification.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**. This setting has no effect unless the temperature differs from one side of the boundary to the other.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	- For a general description of layer and interface selections, see Layer and Interface Selection Tools. - You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.
	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 <i>COMSOL Multiphysics Reference Manual</i>.

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

Lumped System Connector, Interface (Heat Transfer in Shells Interface)

Use this node to connect the distributed finite element model to a thermal network.

The average temperature on the selected interface can be prescribed at a node of the thermal network where an [External Terminal](#) feature of [The Lumped Thermal System Interface](#) is applied.

In return, the heat rate defined at the node of the thermal network is applied on the selected interface.

BOUNDARY SELECTION

Select the boundaries on which to apply the connector.

In addition, select the **Restrict to layered boundaries** checkbox 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** checkbox is not editable.

INTERFACE SELECTION

Set the interfaces for which the connector should be applied. These can be the top and bottom surfaces of the layered material.

Different settings are available, depending on the settings in the [Shell Properties](#) section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, and **Selected interfaces**. With the last option you can select the interfaces for a given layered material, by clearing the checkboxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See [Interface Selections](#) for more details on the interfaces identification.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.
	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 <i>COMSOL Multiphysics Reference Manual</i>.

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 interface.

When the **Isothermal terminal** checkbox is not selected (default), the average temperature on the interface is constrained to the temperature at the terminal node of the thermal network. Select this checkbox to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.



Lumped Composite Thermal Barrier with Shells: Application Library
path **Heat_Transfer_Module/Tutorials,_Thin_Structure/**
lumped_composite_thermal_barrier_shells

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Interfaces > Lumped System Connector, Interface

Heat Transfer in Films > Interfaces > Lumped System Connector, Interface

Heat Transfer in Fractures > Interfaces > Lumped System Connector, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries > Lumped System Connector, Interface

Surface-to-Ambient Radiation, Interface (Heat Transfer in Shells Interface)

Use this node to add surface-to-ambient radiation on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers. The net inward heat flux due to surface-to-ambient radiation is

$$-\mathbf{n} \cdot \mathbf{q} = \varepsilon \sigma (T_{\text{amb}}^4 - T^4)$$

where ε is the surface emissivity, σ is the Stefan-Boltzmann constant (a predefined physical constant), and T_{amb} is the ambient temperature.

BOUNDARY SELECTION

Select the boundaries on which to add surface-to-ambient radiation.

In addition, select the **Restrict to layered boundaries** checkbox 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** checkbox 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 checkboxes 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 <i>COMSOL Multiphysics Reference Manual</i>.
	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 <i>COMSOL Multiphysics Reference Manual</i>.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

SURFACE-TO-AMBIENT RADIATION

Surface Emissivity

The default **Surface emissivity** ε (a dimensionless number between 0 and 1) is taken **From layered material**. For **User defined**, it should be specified. An emissivity of 0 means that the surface emits no radiation at all while an emissivity of 1 means that it is a perfect blackbody.

Ambient Temperature

For **User defined**, enter an **Ambient temperature** T_{amb} . Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

	In 2D, the equation has an additional factor, d_z , to take into account the out-of-plane thickness.
	Theory for Heat Transfer in Thin Structures

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Interfaces > Surface-to-Ambient Radiation, Interface
Heat Transfer in Films > Interfaces > Surface-to-Ambient Radiation, Interface
Heat Transfer in Fractures > Interfaces > Surface-to-Ambient Radiation, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries > Surface-to-Ambient Radiation, Interface

Temperature, Interface (Heat Transfer in Shells Interface)

Use this node to specify the temperature on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers.

BOUNDARY SELECTION

Select the boundaries on which to specify the temperature.

In addition, select the **Restrict to layered boundaries** checkbox to make the node applicable only if a layered material is defined on the boundary. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `slmat1`), otherwise the boundary is marked as not applicable.

Note that when the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Restrict to layered boundaries** checkbox 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 checkboxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See [Interface Selections](#) for more details on the interfaces identification.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.
	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 <i>COMSOL Multiphysics Reference Manual</i>.

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** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	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 checkboxes 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 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 <i>COMSOL Multiphysics Reference Manual</i>.



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.



LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Interfaces > Thermal Contact, Interface

Heat Transfer in Films > Interfaces > Thermal Contact, Interface

Heat Transfer in Fractures > Interfaces > Thermal Contact, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries > Thermal Contact, Interface

Thermal Insulation, Interface (Heat Transfer in Shells Interface)

Use this node to specify thermal insulation on the exterior interfaces of a shell and, for the case of a layered shell, at the interfaces between its layers.

BOUNDARY SELECTION

Select the boundaries on which to specify the thermal insulation. If a layered material (**Material** with **Layer thickness** specified, **Single Layer Material**, **Layered Material Link**, or **Layered Material Stack**) is available, its name is then displayed beside the boundary index (for example, `s1mat1`), otherwise the boundary is marked as not applicable.

INTERFACE SELECTION

Set the interfaces for which the thermal insulation should be prescribed. These can be internal interfaces between layers or the top and bottom surfaces.

Different settings are available, depending on the settings in the [Shell Properties](#) section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by

clearing the checkboxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. See [Interface Selections](#) for more details on the interfaces identification.

- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, a single layer material is defined on the boundary, and the available options in the **Apply to** list are **Top interface**, **Bottom interface**, and **All interfaces**.

You can visualize the selected interfaces by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.
	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 <i>COMSOL Multiphysics Reference Manual</i>.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Interfaces > Thermal Insulation, Interface

Heat Transfer in Films > Interfaces > Thermal Insulation, Interface

Heat Transfer in Fractures > Interfaces > Thermal Insulation, Interface

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Boundaries > Thermal Insulation, Interface

Edge Features

The Heat Transfer interfaces have the following edge nodes and subnodes available:

- Heat Flux (Heat Transfer in Shells Interface)
- Heat Flux (Thin Layer, Thin Film, Fracture)
- Heat Source (Heat Transfer in Shells Interface)
- Line Heat Source
- Lumped System Connector (Heat Transfer in Shells Interface)
- Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)
- Surface-to-Ambient Radiation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Symmetry (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)
- Thermal Insulation, Edge
- 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** checkbox 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** checkbox 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** checkbox.

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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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 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 <i>COMSOL Multiphysics Reference Manual</i>.

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:

Edges > 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** checkbox 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** checkbox 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** checkbox.

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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes for the selection of the individual layers.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is taken **From parent feature** in the **Heat Flux** node. You can change to **User defined** and override the parent node setting with a specific value or expression for L_{th} .

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.

	• For a general description of layer and interface selections, see Layer and Interface Selection Tools. • You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in Using the Extra Dimension Coordinates.
	The desired selection for the node may correspond to boundaries with different layered materials. See Layered Material, Layered Material Link, Layered Material Stack, Layered Material Link (Subnode), and Single Layer Materials in the <i>COMSOL Multiphysics Reference Manual</i>.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Flux** section are defined in the material or spatial frame:

- The **Solid** option specifies that the heat flux q_0 is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See [Conversion Between Material and Spatial Frames](#) for details.

- The default option for the **Heat Flux** subnode is **Nonsolid**, which defines q_0 in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.



This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See [Handling Frames in Heat Transfer](#) and [Material and Spatial Frames](#) for details.

HEAT FLUX

The options available in the **Flux type** list are: **General inward heat flux** (default), **Convective heat flux**, **Nucleate boiling heat flux**, and **Heat rate** (3D components only) button.

- If **General inward heat flux** is selected, it adds q_0 to the total flux across the selected edges. Enter a value for q_0 to represent a heat flux that enters the layer. For example, any electric heater is well represented by this condition and its geometry can be omitted.
- If **Convective heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{\text{ext}} - T)$. 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**. This option is only available when the **Layer type** is **Thermally thin approximation** or **General** in the parent feature.
- If **Nucleate boiling heat flux** is selected, it adds q_0 estimated with the Rohsenow's correlation. See [Nucleate Boiling Heat Flux](#) for the settings. This option is only

available when the **Layer type** is **Thermally thin approximation** or **General** in the parent feature.

- *3D Components:* If **Heat rate** is selected, it adds q_0 in the form $q_0 = P_0/A$ where A is equal to the area of the edge selection. Enter the heat rate P_0 .

	• Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) • Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface) • Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface) • Theory for Heat Transfer in Thin Structures • Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids > Thin Layer > Heat Flux

Heat Transfer in Solids > Thin Film > Heat Flux

Heat Transfer in Porous Media > Fracture > Heat Flux

Ribbon

Physics tab with **Thin Layer**, **Thin Film**, or **Fracture** selected in the model tree:

Attributes > Heat Flux

Heat Source (Heat Transfer in Shells Interface)

This node models a linear heat source (or sink). It adds a heat source $q = Q_b$ or $q = P_b/A$. A positive q is heating and a negative q is cooling.

EDGE SELECTION

Select the edges on which to apply the heat source. The **Restrict to layered edges** checkbox 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** checkbox 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** checkbox.

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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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 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 <i>COMSOL Multiphysics Reference Manual</i>.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Heat Source** section are defined in the material or spatial frame:

- The default option for the **Heat Source** node is **Solid**, which specifies that the heat source Q_b is defined in the material frame. Because the heat transfer variables and equations are defined in the spatial frame, the inputs are internally converted to the spatial frame. See [Conversion Between Material and Spatial Frames](#) for details.
- The **Nonsolid** option defines Q_b in the spatial frame. No frame conversion is needed.
- The **From material** option uses the option selected in the **Material type** list of the **Material Properties** section of the material applied on the domain on which the node is active.



This option has no effect when the component does not contain a moving frame because the material and spatial frames are identical in such cases. With a Deformed Geometry or a Moving Mesh interface, the heat transfer features automatically take into account deformation effects of the material and spatial frames on heat transfer properties. In particular the effects of volume changes on the density are considered. See [Handling Frames in Heat Transfer](#) and [Material and Spatial Frames](#) for details.

BOUNDARY HEAT SOURCE

Select an option from the **Heat Source** list:

For **General source** (default), enter the boundary heat source Q_b . A positive Q_b is heating and a negative Q_b is cooling.

For **Heat rate** enter the heat rate P_b . In this case $Q_b = P_b/A$, where A is the total area of the selected edges (where A is the product of the layer's thickness d_s by the total length of the selected edges).



In 2D components, the equation contains an additional factor, d_z , to take into account the out-of-plane thickness. This is because the selected points correspond to edges in a 3D geometry.

	You can specify the source to be a harmonic perturbation in a frequency domain analysis. To mark the source as being a harmonic perturbation, right-click the node and choose Harmonic Perturbation . Note that this updates the node's icon. See Theory for Harmonic Heat Transfer for more details.
	• Theory for Heat Transfer in Thin Structures • Handling Frames in Heat Transfer

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Heat Source

Heat Transfer in Films > Heat Source

Heat Transfer in Fractures > Heat Source

Ribbon

Physics tab with **Heat Transfer in Shells** selected:

Edges > Heat Source

Line Heat Source

This node models a heat source (or sink) that is so thin that it has no thickness in the model geometry. It is available in 3D on edges. In 2D and 2D axisymmetric, it is available on points.

In theory, the temperature in a line source in 3D is plus or minus infinity (to compensate for the fact that the heat source does not have any volume). The finite element discretization used in COMSOL Multiphysics returns a finite temperature distribution along the line, but that distribution must be interpreted in a weak sense.

MATERIAL TYPE

Select an option in the **Material type** list to specify if the inputs of the **Line Heat Source** section are defined in the material or spatial frame:

- The default option for the **Line Heat Source** node is **Solid**, which specifies that the heat source Q_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

Select an option in the **Heat source** list:

- If **General source** (default) 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** checkbox 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***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 Transfers***support*>**Line Heat Source**
- Heat Transfer in Solids***support*>**Thin Rod**>**Line Heat Source**

Ribbon

Physics tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, or **Bioheat Transfer** selected:

- support* > **Line Heat Source**
- support* > **Thin Rod** > **Line Heat Source**

with *support* as **Egdes** in 3D and **Points** in 2D.

Lumped System Connector (Heat Transfer in Shells Interface)

Use this node to connect the distributed finite element model to a thermal network.

The average temperature on the selected edge can be prescribed at a node of the thermal network where an [External Terminal](#) feature of [The Lumped Thermal System Interface](#) is applied.

In return, the heat rate defined at the node of the thermal network is applied on the selected edge.

EDGE SELECTION

Select the edges on which to apply the connector. The **Restrict to layered edges** checkbox 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** checkbox 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** checkbox.

SHELL PROPERTIES

Different settings are available, depending on the settings in the [Shell Properties](#) section of the parent interface:

- When the **Shell type** is **Layered shell** in the **Shells Properties** section of the parent interface, the same layered material is used in the **Lumped System Connector** node, but you can limit the contribution to individually selected layers by clearing the **Use all layers** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes for the selection of the individual layers.
- When the **Shell type** is **Nonlayered shell** in the **Shells Properties** section of the parent interface, the **Thickness** is **From physics interface** in the **Lumped System Connector** node. You can change to **User defined** and override the interface setting with a specific value or expression for L_{th} .

You can visualize the selected layered materials and layers in each layered material by clicking the **Layer Cross Section Preview** and **Layer 3D Preview** buttons.



- For a general description of layer and interface selections, see [Layer and Interface Selection Tools](#).
- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in [Using the Extra Dimension Coordinates](#).



The desired selection for the node may correspond to boundaries with different layered materials.

See [Layered Material](#), [Layered Material Link](#), [Layered Material Stack](#), [Layered Material Link \(Subnode\)](#), and [Single Layer Materials](#) in the *COMSOL Multiphysics Reference Manual*.

TERMINAL INPUTS

Select an **External Terminal** feature in the **Source** list to define which node of the thermal network provides the heat rate P_{ext} to be prescribed on the edge.

When the **Isothermal terminal** checkbox is not selected (default), the average temperature on the edge is constrained to the temperature at the terminal node of the thermal network. Select this checkbox to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Lumped System Connector

Ribbon

Physics tab with **Heat Transfer in Shells** selected:

Edges > Lumped System Connector

Shell Continuity (Heat Transfer Interface) and Continuity (Heat Transfer in Shells Interface)

Use this node to specify the side-by-side continuity conditions on the temperature field at an edge between two layered materials. By default, the temperature is discontinuous at the interface between two layered materials.

LAYER SELECTION

Select **Source** and **Destination** layered materials from the drop-down lists. This makes the node applicable on the edges adjacent to both selected layered materials. Click the **Layer Cross Section Preview** button to visualize the settings.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	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> 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** checkbox 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** checkbox 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** checkbox.

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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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 <i>COMSOL Multiphysics Reference Manual</i>.

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 (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

This node provides a no flux condition for symmetry edges, similar to a [Thermal Insulation \(Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface\)](#) condition.

It can be applied to exterior edges (3D) or points (2D and 2D axisymmetric) adjacent to the boundaries selected in the parent interface.

EDGE SELECTION

Select the exterior edges on which to apply the symmetry condition. The **Restrict to layered edges** checkbox 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** checkbox 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** checkbox.



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** checkbox.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Symmetry

Heat Transfer in Films > Symmetry

Heat Transfer in Fractures > Symmetry

Ribbon

Physics tab with **Heat Transfer in Shells** selected in the model tree:

Egdes > Symmetry

Temperature (Thin Layer, Thin Film, Fracture, and Heat Transfer in Shells Interface)

Use this subnode to specify the temperature on a set of lines (geometrical edges in 3D or geometrical points in 2D and 2D axisymmetric) that represent the boundaries of a thin domain (layer, film, fracture, or shell). Only edges (3D) or points (2D and 2D axisymmetric) adjacent to the boundaries can be selected in the parent node.

EDGE SELECTION

Select the edges on which to specify the temperature. The **Restrict to layered edges** checkbox 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** checkbox 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** checkbox.

	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>.

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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	You can specify a harmonic variation of the temperature value in a frequency domain analysis of perturbation type, by adding a Harmonic Perturbation subnode. See Theory for Harmonic Heat Transfer for more details.
	• Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) • Thin Film (Heat Transfer Interface) and Fluid (Heat Transfer in Shells Interface) • Fracture (Heat Transfer Interface) and Porous Medium (Heat Transfer in Shells Interface) • Theory for Heat Transfer in Thin Structures
	<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:

Edges > 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 edge condition, this feature is applicable only on external edges; but when added manually, it can be applied on interior edges as well.

At internal edges, and as a subfeature of **Thin Layer**, **Thin Film** or **Fracture** features, the thermal insulation condition can always be applied, but is only taken into account when the **Layer type** is **General** in the parent feature.

At internal edges corresponding to the junction of shells of different layered materials, if the **Layer type** is **General** in the **Thin Layer**, **Thin Film** or **Fracture** parent feature (Heat Transfer interface), or in the **Solid**, **Fluid**, or **Porous Medium** feature (Heat Transfer in Shells interface), thermal insulation is implicit, and the temperature field is discontinuous across the edge, even if no **Thermal Insulation** subnode is active. This default behavior can be modified by using the [Shell Continuity \(Heat Transfer Interface\)](#) and [Continuity \(Heat Transfer in Shells Interface\)](#) feature on the edge.

EDGE SELECTION

Select the edges on which to apply a no flux condition. The **Restrict to layered edges** checkbox 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** checkbox 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** checkbox.

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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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:

Egdes > Thermal Insulation

Thermal Insulation, Edge

Use this node to introduce a temperature discontinuity on edges in 3D components and on points in 2D and 2D axisymmetric components. This feature can only be applied on external edges or points when two or more objects are in contact only along these edge or points.



Radiation in a Cavity. Application Library path **Heat_Transfer_Module/Verification_Examples/cavity_radiation**

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > *support* > Thermal Insulation, Edge

Heat Transfer in Porous Media > Porous Medium > Fluid > *support* > Thermal Insulation, Edge

Heat Transfer in Porous Media > Porous Medium > Porous Matrix > *support* > Thermal Insulation, Edge

Ribbon

Physics tab with **Heat Transfer in Solids and Fluid** or any version of the Heat Transfer interface selected:

***support* > Thermal Insulation, Edge**

Physics tab with **Porous Medium > Fluid** or **Porous Medium > Porous Matrix** selected in the model tree:

Attributes > Thermal Insulation, Edge

with *support* as **Egdes** in 3D and **Points** in 2D.

Thin Rod

Use this node to define the thermal and radius properties of conductive rods located on edges in a 3D component.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Volume Reference Temperature

This section is available when a temperature-dependent density defined in a material is used. On the material frame, the density is evaluated in relation to a reference temperature in order to ensure conservation of the mass in the presence of temperature variations. By default the **Common model input** is used. This corresponds to the variable `minput.Tempref`, which is set to 293.15 K by default. To edit it, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Volume reference temperature** in the **Expression for remaining selection** section.

The other options are **User defined** and all temperature variables from the physics interfaces included in the model.



This model input does not override the **Reference temperature** T_{ref} set in the **Physical Model** section of the physics interface, which is used to evaluate the reference enthalpy, and a reference density for incompressible nonisothermal flows.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the parent interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

THIN ROD

The **Rod radius** r_1 should be specified.

HEAT CONDUCTION

The **Thermal conductivity** k_1 should be specified. By default it is taken **From material**. For **User defined** select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** to enter another value or expression.

THERMODYNAMICS

By default the **Density** ρ_1 and the **Heat capacity at constant pressure** $C_{p,1}$ values are taken **From material**. See [Material Density in Features Defined in the Material Frame](#) if a temperature-dependent density should be set. For **User defined** enter other values or expressions.



These additional subnodes are available for the **Thin Rod** node:

- [Line Heat Source](#) — to add an internal heat source, Q_1 , within the rod.
- [Temperature \(Thin Rod\)](#) — to set a prescribed temperature condition on a specified set of points.
- [Point Heat Flux \(Thin Rod\)](#) — to add a heat flux through a specified set of points.
- [Surface-to-Ambient Radiation \(Thin Rod\)](#) — to add surface-to-ambient radiation at the rod end points.

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids and Fluids > Edges > Thin Rod

Heat Transfer in Solids > Edges > Thin Rod

Heat Transfer in Fluids > Edges > Thin Rod

Heat Transfer in Porous Media > Edges > Thin Rod

Bioheat Transfer > Edges > Thin Rod

Ribbon

Physics tab with **Heat Transfer**, **Heat Transfer in Solids**, **Heat Transfer in Fluids**, **Heat Transfer in Porous Media**, **Heat Transfer in Building Materials**, or **Bioheat Transfer** selected:

Edges > Thin Rod

Point Features

The Heat Transfer interfaces have the following point nodes and subnodes available:

- Point Heat Flux (Thin Rod)
- Point Heat Source
- Point Heat Source on Axis
- Surface-to-Ambient Radiation (Thin Rod)
- Temperature (Thin Rod)



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

Point Heat Flux (Thin Rod)

Use this subnode to add heat flux at points of a thin rod. A positive heat flux adds heat to the rod.

HEAT FLUX

The available options in the **Flux type** list are: **General inward heat flux** (default), **Convective heat flux**, and **Nucleate boiling heat flux**.

- If **General inward heat flux** is selected, it adds q_0 to the total flux across the selected points. Enter a value for q_0 to represent a heat flux that enters the rod.
- If **Convective heat flux** is selected, it adds q_0 in the form $q_0 = h \cdot (T_{\text{ext}} - T)$. Enter the **Heat transfer coefficient** h and the **External temperature** T_{ext} . The value depends on the geometry and the ambient flow conditions.
- If **Nucleate boiling heat flux** is selected, it adds q_0 estimated with the Rohsenow's correlation. See [Nucleate Boiling Heat Flux](#) for details about the settings.



Thin Rod

LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Solids > Thin Rod > Point Heat Flux

More locations are available. For example:

Heat Transfer in Fluids > Thin Rod > Point Heat Flux

Ribbon

Physics tab with **Thin Rod** selected in the model tree:

Attributes > Point Heat Flux

Point Heat Source

This node, available for 3D components, models a heat source (or sink) that is so small that it can be considered to have no spatial extension.

In theory, the temperature in a point source in 3D is plus infinity (to compensate for the fact that the heat source does not have a spatial extension). The finite element discretization used in COMSOL Multiphysics returns a finite value, but that value must be interpreted in a weak sense.

POINT HEAT SOURCE

Enter the **Point heat source** Q_p in unit power. A positive Q_p corresponds to heating while a negative Q_p corresponds to cooling.

HEAT SOURCE RADIUS

With the Heat Transfer Module, you can model the heat source explicitly and apply it on a ball or disk around the point.

Select the **Specify heat source radius** checkbox 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
	<i>Heat Conduction with a Localized Heat Source on a Disk: Application</i> 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** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	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 node available:

- [Thermal Connection](#)

The Surface-to-Surface Radiation and the Orbital Thermal Load interfaces have the following global nodes available:

- [Events Timeline](#)
- [External Radiation Source](#)
- [Generate Events Interface](#)
- [Ground Pointing](#)
- [Implicit Event](#)
- [Orbital Parameters](#)
- [Planet Properties](#)
- [Spacecraft Axes](#)
- [Spacecraft Orientation](#)
- [Sun Properties](#)
- [Symmetry for Surface-to-Surface Radiation](#)

The Heat Transfer in Shells interfaces have the following global node available:

- [Thermal Connection, Interface-Interface](#)

The Lumped Thermal System interface has the following global nodes available:

- Conductive Thermal Resistor
- Convective Thermal Resistor
- External Terminal
- Heat Pipe
- Heat Rate
- Heat Rate Source
- Nucleate Boiling Heat Rate
- Radiative Heat Rate
- Radiative Thermal Resistor
- Subsystem Definition
- Subsystem Instance
- Temperature
- Thermal Capacitor
- Thermal Mass
- Thermoelectric Module



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

Thermal Connection

Use this node to connect two boundary selections by a thermal resistor, thermal capacitor or **Lumped Thermal System** interface.

BOUNDARY SELECTION, CONNECTOR 1

Select the boundaries corresponding to the first connector.

BOUNDARY SELECTION, CONNECTOR 2

Select the boundaries corresponding to the second connector.

THERMAL CONNECTION

Select the **Connection type** from the list: **Conductive thermal resistor** (default), **Thermal capacitor**, or **Lumped thermal system**.

Conductive Thermal Resistor

It models heat loss by conduction in a domain by a thermal resistance R (SI unit: K/W) between the two connectors. This resistance 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 Thermal Connection](#) for more details on the underlying theory.

- When **Specify** is **Thermal resistance** (default), 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** checkbox 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** checkbox 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** checkbox is selected in the **Thermal Connection** 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 **Thermal Connection** 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 between the connectors 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 is used. If the material properties are temperature-dependent, they are evaluated at the average temperature of the connectors.



Equivalent Thermal Conductivity Correlations

Optically Thick Participating Medium

This section is available when the **Optically thick participating medium** checkbox is selected in the **Thermal Connection** 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

$$k_R = \frac{16n_r^2\sigma(T_{avg})^3}{3\beta_R}$$

where n_r is the refractive index (dimensionless), σ is the Stefan-Boltzmann constant (SI unit: $\text{W}/(\text{m}^2 \cdot \text{K}^4)$), T_{avg} is the average temperature of the connector 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 **Thermal Connection** section is used. If the material properties are temperature-dependent, they are evaluated at the average temperature of the connector.

Thermal Capacitor

It models heat storage in a domain with a thermal capacitance C (SI unit: J/K) between the two connectors.

For a steady-state problem the temperature difference does not change with time and the imposed heat flux simplifies to zero.

See [Theory for Thermal Connection](#) for more details on the underlying theory.

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 pressure**, 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 pressure**, C_p . Select any material from the **Material** list to define C_p **From material**. For **User defined** enter a value or expression.

Lumped Thermal System

It models a thermal network defined by a **Lumped Thermal System** interface between the two connectors.

The average temperature on the selected boundaries for each connector 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 of each connector.

For each connector, 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. The initial temperature of the connector is also taken from the selected **External Terminal** feature for a transient simulation.

When the **Isothermal terminal** checkbox 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 checkbox to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.

	When only one External Terminal feature is selected in Connector 1 or Connector 2 , the behavior is equivalent to the Lumped System Connector feature.
	Theory for Thermal Connection
	<i>Lumped Composite Thermal Barrier</i> : 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 > Thermal Connection

Ribbon

Physics tab with **Heat Transfer in Solids and Fluids** or any version of the Heat Transfer interface selected:

Global > Thermal Connection

Events Timeline

The **Events Timeline** feature is present by default. Additional **Events Timeline** features can be added to the **Orbital Thermal Loads** interface in order, for example, to compare different scenarios based on different events. Only one active **Events Timeline** feature is expected when solving with a **Orbital Thermal Loads** physics interface. The **Events Timeline** feature allows to manage the implicit and explicit built-in events to consider during the spacecraft orbital motion.

INITIAL ORIENTATION

Select the **Fast tumbling** checkbox to enable the fast tumbling approximation at the beginning of the simulation. The approximation remains active until an event that specifies that the approximation is not longer enabled is triggered. This approximation replaces the evaluation of the radiation received by the Sun at every position by an averaged value. This option can be used if you assume that the spacecraft is tumbling sufficiently fast so that all environmental heat loads can be averaged. It saves computational time by not computing the external sources view factors.

In the **Axes feature** list, select the **Spacecraft Axes** feature specifying the initial pointing direction of the spacecraft.

Set the **Orientation feature** list to the **Spacecraft Orientation** feature specifying the initial orientation of the spacecraft.

IMPLICIT EVENTS

The table lists all the implicit events taken into account during the spacecraft orbit. For each event set the **Event type** to any implicit available event.

Set the **Axes feature** and the **Orientation feature** to define the spacecraft positioning when the event occurs. The positioning is repeated each time the event type is encountered.

Set the **Fast tumbling** checkbox to control whether the fast tumbling approximation (see [Initial Orientation](#)) is active or not when the event occurs.

There are four types of predefined implicit events:

- **Into eclipse:** triggered when the spacecraft enters the eclipse. This event is always available.
- **Out of eclipse:** triggered when the spacecraft exits the eclipse. This event is always available.
- **Implicit event:** triggered when a user-defined condition is fulfilled. This event is optional, multiple **Implicit event** features can be added in the interface.
- **Ground station:** triggered when the ground station is visible from the spacecraft given the approach and elevation angles. Another event is triggered when the ground station is out-of-sight. The fast tumbling approximation is always disabled when the ground station becomes visible. When the station becomes out-of-sight the approximation is enabled or not depending on the corresponding setting in the **Ground station** feature. This event is optional, multiple **Ground station** features can be added in the interface.

EXPLICIT EVENTS

Add events in the **Explicit events** table by setting at which **Time** the event occurs and what is its corresponding **Axes feature** and the **Orientation feature** at this time. The event is repeated every period specified in the **Period of event** column. Set **Period of event** to **inf** (default) if the event does not repeat. Set the **Fast tumbling** checkbox to control whether the fast tumbling approximation (see [Initial Orientation](#)) is active or not when the event occurs.



See [Modeling of Fast Tumbling](#) for more details about the fast tumbling definition of the heat loads.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Events Timeline

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Events Timeline

External Radiation Source

Use this node in 2D and 3D components to define an external radiation source as a point or directional radiation source with view factors calculation. Only the external view factors (F_{ext}) are recomputed when the radiation source moves, not the complete view factors. When specular radiation is involved (with **Opaque Surface**, **Specular Mirror**, or **Semitransparent Surface**), the cost of computation of these external view factors can be important. Each **External Radiation Source** node contributes to the incident radiative heat flux on all spectral bands, G_{B_i} on all the boundaries where a **Diffuse Surface**, **Diffuse Mirror**, **Opaque Surface**, **Specular Mirror**, 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** checkbox 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**, **Specular Mirror**, 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**, **Specular Mirror**, 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)**, **Meteorological data (ASHRAE 2017)**, or **Meteorological data (ASHRAE 2021)** 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)** checkbox 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)** checkbox 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 checkbox 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** checkbox is selected, and the time is then automatically updated with the time from the solver. Clear this checkbox 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_g as the incident radiative intensity coming directly from the Sun. I_g 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_g . 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 $q_{s,i} = q_{0,s} \text{FEP}_i(T_{\text{sun}})$ where $\text{FEP}_i(T_{\text{sun}})$ is the fractional blackbody emissive power over B_i interval at $T_{\text{sun}} = 5780 \text{ K}$.

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 $P_{s,i} = \text{FEP}_i(T_s)P_s$ where $\text{FEP}_i(T_s)$ 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 $q_{s,i} = \text{FEP}_i(T_s)q_{0,s}$ where $\text{FEP}_i(T_s)$ 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
	<i>Sun's Radiation Effect on Two Coolers Placed Under a Parasol:</i> 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



The **Generate Events Interface** feature is obsolete and cannot be added in version 6.2 and later, but may be present in models created by earlier versions of the software.

You are advised to update your model to use the [Events Timeline](#) feature instead.

The **Generate Events Interface** feature is present by default. Additional **Generate Events Interface** features can be added to the **Orbital Thermal Loads** interface in case, for example, to compare different scenarios based on different events. The **Generate Events Interface** feature controls an events interface that is solved for with the **Orbital Thermal Loads** interface in order to detect the events times. It also automatically adds explicit events when the spacecraft goes into and out of eclipse.

GENERATE EVENTS INTERFACE

Click the **Create New** button to add a new **Events Interface** in the same component with the events from the **Implicit Event** and **Explicit Event** sections. Alternatively, select an existing event interface in the **Events interface** list and click **Refresh** button to overwrite the autogenerated features of the selected event interface and replace them with the ones from the **Implicit Event** and **Explicit Event** sections. Before removing the autogenerated features, the user is prompted with a dialog to confirm this action. The manually added features of the events interface are untouched when the **Refresh** button is clicked.

INITIAL ORIENTATION

Select the **Fast tumbling** checkbox to enable the fast tumbling approximation at the beginning of the simulation. The approximation remains active until an event that specifies that the approximation is not longer enabled is triggered. This approximation replaces the evaluation of the radiation received by the Sun at every position by an averaged value. This option can be used if you assume that the spacecraft is tumbling sufficiently fast that all environmental heat loads can be averaged. It saves computational time by not computing the external sources view factors.

In the **Axes feature** list, select the **Spacecraft Axes** feature specifying the initial pointing direction of the spacecraft.

Set the **Spacecraft Orientation** list to the **Spacecraft Orientation** feature specifying the initial orientation of the spacecraft.

IMPLICIT EVENTS

The table lists all the implicit events taken into account during the spacecraft orbit. For each event set the **Event type** to any implicit available event.

Set the **Axes feature** and the **Orientation feature** to define the spacecraft positioning when the event occurs. The positioning is repeated each time the event type is encountered.

Set the **Fast tumbling** checkbox to control whether the fast tumbling approximation (see [Initial Orientation](#)) is active or not when the event occurs.

There are four types of predefined implicit events:

- **Into eclipse:** triggered when the spacecraft enters the eclipse. This event is always available.
- **Out of eclipse:** triggered when the spacecraft exits the eclipse. This event is always available.
- **Implicit event:** triggered when a user-defined condition is fulfilled. This event is optional, multiple **Implicit event** features can be added in the interface.
- **Ground station:** triggered when the ground station is visible from the spacecraft given the approach and elevation angles. Another event is triggered when the ground station is out-of-sight. The fast tumbling approximation is always disabled when the ground station becomes visible. When the station becomes out-of-sight the approximation is enabled or not depending on the corresponding setting in the **Ground station** feature. This event is optional, multiple **Implicit event** features can be added in the interface.

EXPLICIT EVENTS

Add events in the **Explicit events** table by setting at which **Time** the event occur and what is its corresponding **Axes feature** and the **Orientation feature** at this time. The event is repeated every period specified in the **Period of event** column. Set **Period of event** to **inf** (default) if the event does not repeat. Set the **Fast tumbling** checkbox to control

whether the fast tumbling approximation (see [Initial Orientation](#)) is active or not when the event occurs.



See [Modeling of Fast Tumbling](#) for more details about the fast tumbling definition of the heat loads.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Generate Events Interface

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Generate Events Interface

Ground Pointing

The **Ground Pointing** feature is designed to point toward a ground station during the orbit. It is possible to define multiple **Ground Pointing** features.

GEOGRAPHIC POSITION

Select an option from the **Position defined by** list: **Coordinates** (default) or any **Ambient Properties** defined under **Definitions** and with **Ambient data** set to **Meteorological data (ASHRAE 2013)**, **Meteorological data (ASHRAE 2017)**, or **Meteorological data (ASHRAE 2021)**. When selected from this list, the position of the ground station is set to the **Weather station** selected in the **Ambient Properties** node under **Definitions**.

If **Coordinates** is selected, define the **Latitude** and **Longitude** (SI unit radians, default value 0 deg) of the ground station.

POINTING TRIGGER

Selected any of the available **Event indicator** options to define the criteria that defines when the spacecraft begins to be on target and when it goes out of target.

When **Elevation angle** is selected enter the value for the **Threshold** (SI unit radians, default value 0 deg) to define the on target and out of target positions.

When **Ranging** is selected, define the **Threshold** (SI unit m, default value 0 km) that corresponds to the distance to the target that defines the limit between the on target and out target regions.

Use the **User defined** option to enter an expression for the **Event indicator** and for the **Threshold**.

For all types of **Event indicator** you can select the **Point to target for a specified time** checkbox. When this is selected the **Axes and Orientation out of Target** settings are applied at the end of the specified period instead of when the trigger condition is no longer verified.

AXES AND ORIENTATION ON TARGET

The **Primary axis orientation** is automatically defined from the ground point direction. Define the **Secondary axis orientation** from the corresponding checkbox.

The **Rotations** about the primary, secondary, and tertiary axis can be either set to **None**, **Angular position** (SI unit radians, default value 0 deg), or **Angular rate** (SI unit radians per second, default value 0 deg/s). Note that the fast tumbling option is always disabled when the ground station is on target.

AXES AND ORIENTATION OUT OF TARGET

When the **Event indicator** is no longer active or the **Pointing time** is elapsed, the spacecraft is no longer pointing to the station. Set the **Fast tumbling** checkbox to control whether the fast tumbling approximation (see [Initial Orientation](#)) is active or not when the spacecraft goes out of target. Select an **Axes feature** and a **Orientation feature** to define the out of target spacecraft pointing and orientation.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Ground Pointing

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Ground Pointing

Implicit Event

The **Implicit Event** feature is designed to specify user defined implicit events which are compatible with the pointing and orientation framework. It is possible to define multiple **Implicit Event** features.

Once added to the physics interface, any **Implicit Event** feature can be selected in the **Event type** column of the **Events Timeline** feature.

Set an expression for the **Event indicator**, V , that indicates the global quantity that should be monitored. Select either **Upper threshold** or **Lower threshold** and set the **Threshold** value, V_{th} , to define when the implicit event is triggered.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Implicit Event

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Implicit Event

Orbital Parameters

This feature contains all the information needed to define the spacecraft orbit.

Select an **Orbit type** among **Elliptical** (default), **Equatorial elliptical**, **Circular**, **Equatorial circular**, and **User defined**.

Elliptical is the most general option. It defines the orbit of the spacecraft from the six Keplerian elements as in [Figure 6-4](#):

- **Semimajor axis**, a (SI unit m, default value $ot1.R_{planet}+1000$ [km] assuming $ot1$ is the interface name),
- **Eccentricity**, e (dimensionless, default value 0) which is a value between 0 (included) and 1 (excluded),
- **Inclination**, i (SI unit radians, default value 0 deg), which is a positive angle between 0 and 180°,
- The ascending node can be defined either from the **Longitude of ascending node**, Ω (SI unit radians, default value 0 deg), or from the **Local time at ascending node**, t_{Ω} (SI unit seconds, default value 0 h)

- **Argument of periapsis**, ω (SI unit radians, default value 0 deg), and
- **True anomaly at start time**, v_0 (SI unit radians, default value 0 deg).

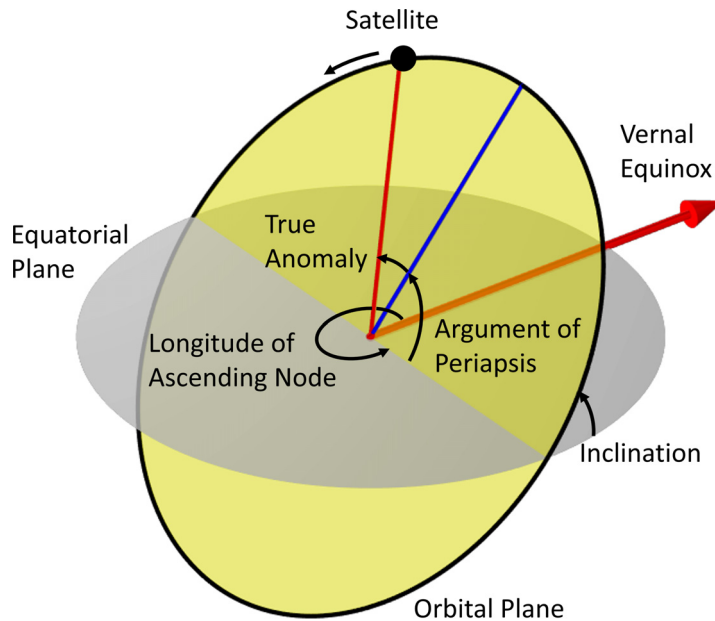


Figure 6-4: Definition of Keplerian elements.

Equatorial elliptical is similar to the **Elliptical** option with **Inclination**, i , and the **Longitude of Ascending Node**, Ω , are zero. In this configuration the **Argument of periapsis** is referred as **Longitude of periapsis**.

The **Circular** option is another option with four orbital parameters. The **Eccentricity**, e , and the **Argument of periapsis**, ω , are zero. In this configuration the **Radius**, R , is equivalent to the semi-major axis, and the **True anomaly at start time**, v_0 is referred as the **Argument of latitude at start time**.

The **Equatorial circular** choice is the simplest option with only two parameters: the **Radius**, R , and **True anomaly at start time**, v_0 . true longitude (SI unit rad, default value 0 deg).

The **User defined** option allows to freely define the position of the spacecraft in the equatorial coordinate system based. Select a **Spacecraft position source** among **Functions** (default) and **Local Table**.

- When **Functions** is selected, each coordinate is defined as a function of time. For **User defined**, enter a value or expression. Else, select a function defined under **Global Definitions** or **Definitions**. Only those which argument and result units are respectively relative to time and length are available.
- When **Local table** is selected, a table appears, where time and corresponding spacecraft coordinates can be specified or imported from a text file by clicking the **Load from File** (📁) button under the table. Units can be specified for the **Time** and the **Position**. These data are then interpolated using a piecewise cubic method, and extrapolated linearly.



With **User defined** selected as **Orbit type**, orbit period is not defined and the **Output times specification** of [Orbit Calculation](#), [Orbital Temperature](#), and [Orbit Thermal Loads](#) studies must be set to Time steps.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Orbital Parameters

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Orbital Parameters

Planet Properties

The **Planet Properties** node contains the parameters for the definition of the mass, radius, and radiative properties.

PLANET PROPERTIES

The **Planet Properties** list is set to **Earth** by default. Selecting this option is equivalent to the **User defined** option with the following settings ([Ref. 48](#)):

- Mass, $M = 5.972e24[\text{kg}]$
- Radius, $R = 6378[\text{km}]$

- Tilt angle, $\varepsilon_{\text{tilt}} = 23.45[\text{degC}]$
- Rotation rate, $\Omega = 1/24[\text{h}]$

The position of the planet at $t = 0$ s can be defined from different parameters depending on the selected option for **Planet longitude at start time**. When **Determined from GMT at start time is selected**, define the Greenwich Mean Time (GMT) that correspond to the planet position at $t = 0$ s. Select **Longitude at subspacecraft point** to define the initial planet position from the spacecraft position at $t = 0$ s. Choose **Longitude facing Vernal point** to define the initial planet position from the position of the Sun at $t = 0$ s.

RADIATIVE PROPERTIES

Set **Geographic position dependence** to **Constant** when the radiative properties are assumed to be uniform all around the planet. In case they depend on the location on the planet, select the **Latitude and longitude** option.

When **Geographic position dependence** is set to **Constant**, the **Albedo** and **Planet radiative flux** can be defined as **None** (default), **User-defined for each band**, or **User-defined distribution**.

When **User-defined distribution** option is selected for **Albedo** or **Planet radiative flux**, the expression is assumed to be a function of `otl.lambda`, where `otl` is the name of the physics interface. This distribution is integrated on each spectral band to obtain the albedo and planet radiative flux for each spectral band.

When **Albedo** or **Planet radiative flux** is set to **User defined for each band**, enter a value for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength.

When **Geographic position dependence** is set to **Latitude and longitude**, the **Albedo** and **Planet radiative flux** can be defined as **None** (default), or **User defined for each band**.

When **User defined distribution** option is selected for **Albedo** or **Planet radiative flux**, select a two argument function to define the corresponding property for each spectral band.

If the **Wavelength dependence of radiative properties** is set to **Constant** in the interface node, then the radiative properties are given as expressions or latitude and longitude functions that do not depend on the wavelength `otl.lambda`.

The planet is approximated via a set of radiation sources on a spherical cap representing the part of the planet that can be seen from the spacecraft. Under **Planet discretization**,

set the **Number of rings** (default value 5) and the number of **Points per ring** (default value 10) to define the total number of points that should be used to discretize the planet and how they should be placed. The total number of points used is $N = 1 + N_m N_n$, where N_m is the number of ring and N_n is the number of points per ring. If the radiative properties of the planet depend on the geographic position and an accurate interpretation of the data is expected, the number of points can be increased. Increasing the total number of points N , also increases the computational time.



See [Modeling the Planet](#) for more details about the planet discretization.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Planet Properties

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Planet Properties

Spacecraft Axes

The spacecraft axes are used to define the primary and secondary, and consequently the third, axes of the spacecraft coordinate system orientation. This is a cartesian coordinate system with its origin at the location of the spacecraft in orbit. Multiple **Spacecraft Axes** features can be defined. They are then available in the **Events Timeline** features.

Set the **Primary axis** to **From primary selection**, **+X direction**, **+Y direction**, or **User defined** to define the primary pointing direction. When **From primary selection** is selected, the primary pointing direction is defined as the normalized averaged normal vector over the selected surfaces. When **User defined** is selected, the primary point direction is defined as the user defined vector divided by its norm.

Similar user inputs are available to define the **Secondary axis**.

The tertiary axis is then defined as the normalized cross product of the primary and secondary axis.

If the primary and secondary axis are collinear an error is triggered. In case the secondary vector is not a unit vector perpendicular to the primary axis, the actual secondary axis is the normalized projection of the secondary vector onto the plane normal to the primary axis.



See [Theory for Orbital Thermal Loads](#) for the definition of the spacecraft position over time and the coordinate system transformation.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Spacecraft Axes

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Spacecraft Axes

Spacecraft Orientation

The **Spacecraft Orientation** feature is designed be combined with a **Spacecraft Axes** feature to define the spacecraft positioning. Multiple **Spacecraft Orientation** features can be defined. They are then available in the **Events Timeline** features.

The spacecraft positioning is set so that it faces exactly the primary pointing direction, and is rotated such that the secondary spacecraft pointing direction points toward the projection of the second orientation direction onto the plane normal to the primary axis.

Select one of the following option to define the **Primary axis orientation**:

- **Nadir** (default) to make the primary spacecraft direction pointing toward the center of the planet, the opposite of **Zenith**.
- **Zenith** to make the primary spacecraft direction pointing away from the center of the planet.
- **Sun** or **Anti-Sun** to make the primary spacecraft direction pointing toward the Sun or to the opposite direction.
- **Velocity** or **Antivelocity** to make the primary spacecraft direction pointing toward the direction of motion of the spacecraft or to the opposite direction.

- **Normal** or **Antinormal** to make the primary spacecraft direction pointing toward the normal of the orbital plane or to the opposite direction.



With **User defined** selected as **Orbit type** in the **Orbital Parameters** feature, the normal axis is defined as the normalized cross product of the zenith axis and the spacecraft velocity axis in the equatorial coordinate system.

- **Celestial** to make the primary spacecraft direction pointing toward a point by giving its celestial coordinates. Set the **Declination** and **Right ascension** to locate the point with its celestial coordinates.
- **User defined** to define explicitly the pointing direction.

Similar options are available to define the **Secondary pointing** direction.

The **Primary pointing** and the **Secondary pointing** directions should not be collinear.



See [Theory for Orbital Thermal Loads](#) for the definition of the spacecraft position over time and the coordinate system transformation.

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Spacecraft Orientation

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Spacecraft Orientation

Sun Properties

The **Sun Properties** feature defines the Sun position and the solar flux.

Select any of the predefined option for the **Sun direction** (**Winter solstice**, **Summer solstice**, **Spring equinox**, or **Fall equinox**) to define the Sun vector and solar flux. In this case the **Sun vector in ECS field** is not editable and show the vector definition for the selected value.

When the **Solar flux** is set to **Automatic** a Planck's distribution for a blackbody at $T_s = 5780$ K is used for the radiation in case of multiple spectral bands. The solar flux intensity (SI unit W/m^2) is defined according to [Table 6-6](#).

TABLE 6-6: SOLAR FLUX INTENSITIES.

SUN DIRECTION	SOLAR FLUX INTENSITY
Winter solstice	1414 [W/m^2]
Summer solstice	1322 [W/m^2]
Spring equinox	1367 [W/m^2]
Fall equinox	1367 [W/m^2]

The **Solar flux** can also be set to **User defined** and **User defined per band** in case of multiple spectral bands. In case of gray surface model, only **User defined** is available.

When the **Sun direction** is set to **User defined**, the **Sun vector in ECS field** is editable. The **Solar flux** can be defined similarly as above except that the **Automatic** option is not available.

When the **Solar flux** is set to **User defined for each band**, enter a value for the **Solar flux**, $q_{0s,i}$ (SI unit W/m^2 , default value $0 \text{ W}/\text{m}^2$) for each spectral band. Within a spectral band, each value is assumed to be independent of wavelength.

When **Solar flux** is set to **User defined**, define $q_{0s,\lambda}$ (SI unit W/m^3 , default value $0 \text{ W}/\text{m}^3$). The wavelength may be accessed through the `ot1.lambda` variable. This distribution is integrated on each spectral band to obtain the source heat flux $q_{0s,i}$ for each spectral band.

Any **Clear sky noon beam normal irradiance** defined from an **Ambient properties** node can also be selected to define q_{0s} . In this case spectral the bands the solar flux distribution per band is defined following Planck's distribution for a blackbody at T_s .

LOCATION IN USER INTERFACE

Context Menus

Orbital Thermal Loads > Sun Properties

Ribbon

Physics tab with **Orbital Thermal Loads** selected:

Global > Sun Properties

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 6-7](#) summarizes the available options for each dimension.

TABLE 6-7: OPTIONS FOR SYMMETRY DEFINITION, EXAMPLE WITH 5 SECTORS.

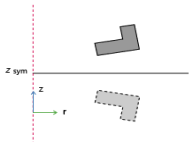
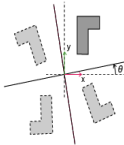
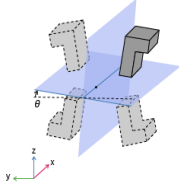
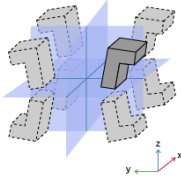
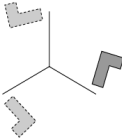
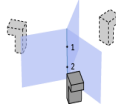
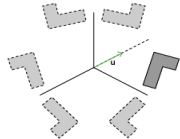
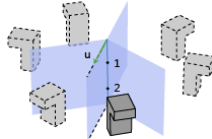
SYMMETRY TYPE	2D	2D AXISYMMETRIC	3D
One Plane			
Two Planes		Not applicable	
Three Planes	Not applicable	Not applicable	

TABLE 6-7: 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 symmetry** (default), **Two perpendicular symmetry planes**, or **Sector symmetry**.
- In 2D axisymmetric components, only the **Plane symmetry** option is available.
- In 3D components, select between **Plane symmetry** (default), **Two perpendicular symmetry planes**, **Three perpendicular symmetry planes**, or **Sector symmetry**.

Depending on the **Type of symmetry** selected from the list and on the dimension, further settings are required.

When **Apply symmetry to external radiation sources** is selected (default), all external radiation sources are duplicated with respect to the symmetry condition. Otherwise, the external radiation sources do not follow the symmetry condition.

Plane 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 defining reflection plane** and of the **Second point defining reflection plane**. Else, if the **Selection method** is

Point selection, you can directly select the points from the **Graphics** window. In the **First Point Defining Reflection Plane** and **Second Point Defining Reflection Plane** 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 **Reflection plan position**, z_{sym} . Else, if the **Selection method** is **Point selection**, you can directly select the point from the **Graphics** window. In the **First Point Defining Reflection Plane** section, first use the **Active** button to toggle between turning ON  and OFF  selection, that is, making the selection active. 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 defining reflection plane**, the **Second point defining reflection plane**, and the **Third point defining reflection plane**. Else, if the **Selection method** is **Point selection**, you can directly select the points from the **Graphics** window. In the **First Point Defining Reflection Plane**, **Second Point Defining Reflection Plane**, and **Third Point Defining Reflection Plane** 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 symmetry plane** checkbox is selected by default. Clear it to hide the plane in the **Graphics** window.

Two Perpendicular Symmetry Planes

- In 2D components, the planes intersection line is along the out-of-plane direction. The **Point at intersection of symmetry planes** and the **Rotation angle** from Cartesian axis, θ , should be defined. If the **Selection method** is **Coordinates**, set the **x** and **y** coordinates of the **Point at intersection of symmetry planes**. Else, if the **Selection method** is **Point selection**, you can directly select the point from the **Graphics** window. In the **Point at Intersection of Symmetry Planes** section, first use the **Active** button to toggle between turning ON  and OFF  selection, that is, making the selection active. Then select the point from the **Graphics** window.
- 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 the intersection of symmetry planes** and the **Rotation angle** from Cartesian axis, θ , should be defined. If the **Selection method** is **Coordinates**, set the **x**, **y**, and **z** coordinates of the **Point at intersection of symmetry planes**. Else, if the **Selection method** is **Point selection**, you can directly select the point from the **Graphics** window. In the **Point at Intersection of Symmetry Planes** section,

first use the **Active** button to toggle between turning ON  and OFF  selection, that is, making the selection active. Then select the point from the **Graphics** window.

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 in the figure displayed below the settings, for each case of alignment of the intersection line.

The **Show symmetry planes** checkbox is selected by default. Clear it to hide the planes in the **Graphics** window.

Three Perpendicular Symmetry Planes

The planes are parallel to the Cartesian axis. Only the **Point at intersection of symmetry planes** should be defined. If the **Selection method** is **Coordinates**, set the **x**, **y**, and **z** coordinates of the **Point at intersection of symmetry planes**. Else, if the **Selection method** is **Point selection**, you can directly select the point from the **Graphics** window. In the **Point at Intersection of Symmetry Planes** section, first use the **Active** button to toggle between turning ON  and OFF  selection, that is, making the selection active. Then select the point from the **Graphics** window.

The **Show symmetry planes** checkbox is selected by default. Clear it to hide the planes in the **Graphics** window.

Sector 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. If the **Selection method** is **Coordinates**, set the **x** and **y** coordinates of the **Point defining central symmetry**. Else, if the **Selection method** is **Point selection**, you can directly select the point from the **Graphics** window. In the **Point Defining Central Symmetry** section, first use the **Active** button to toggle between turning ON  and OFF  selection, that is, making the selection active. Then select the point from the **Graphics** window.
- In 3D components, the symmetry axis is defined by two points. If the **Selection method** is **Coordinates**, set the **x**, **y**, and **z** coordinates of the **First point defining sector symmetry axis** and of the **Second point defining sector symmetry axis**. Else, if the **Selection method** is **Point selection**, you can directly select the points from the **Graphics** window. In the **First Point Defining Sector Symmetry Axis** and **Second Point Defining Sector Symmetry Axis** 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.

Enter a value for the **Number of sectors**. This should be a numerical value greater or equal to 2.

The **Show symmetry axis** checkbox is selected by default. Clear it to hide the axis in the **Graphics** window.

If the **Reflection for symmetrical sector** checkbox 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

Thermal Connection, Interface-Interface

Use this node to connect the interfaces of two boundary selections by a thermal resistor, thermal capacitor or **Lumped Thermal System** interface.

BOUNDARY SELECTION, CONNECTOR I

Select the boundaries corresponding to the first connector.

BOUNDARY SELECTION, CONNECTOR 2

Select the boundaries corresponding to the second connector.



The selections are only applicable on boundaries with **Layer type** set to **General** in the [Thin Layer \(Heat Transfer Interface\)](#) and [Solid \(Heat Transfer in Shells Interface\)](#), [Thin Film \(Heat Transfer Interface\)](#) and [Fluid \(Heat Transfer in Shells Interface\)](#), and [Fracture \(Heat Transfer Interface\)](#) and [Porous Medium \(Heat Transfer in Shells Interface\)](#) features. In addition, the layered material applied on the boundaries must correspond to the one set in the **Shell Properties** section.

SHELL PROPERTIES

Set the layered material on which each connector can be applied.

INTERFACE SELECTION

Set the interfaces for which each connector should be applied. The available options in the **Connector 1, apply to** and **Connector 2, apply to** list are **Top interface** (default), **Bottom interface**, and **Exterior interfaces**. The later is not available when both connectors share a boundary and the same layered material defined in the **Shell properties** section.



- For a general description of layer and interface selections, see [Layer and Interface Selection Tools](#).
- You can provide material parameters with a through-thickness variation by using expressions containing the extra dimension coordinate as described in [Using the Extra Dimension Coordinates](#).



The desired selection for the node may correspond to boundaries with different layered materials.

See [Layered Material](#), [Layered Material Link](#), [Layered Material Stack](#), [Layered Material Link \(Subnode\)](#), and [Single Layer Materials](#) in the *COMSOL Multiphysics Reference Manual*.



Upside and downside settings can be visualized by plotting the global normal vector (nx, ny, nz), that always points from downside to upside. Note that the normal vector (ht.nx, ht.ny, ht.nz) may be oriented differently.

See [Tangent and Normal Variables](#) in the *COMSOL Multiphysics Reference Manual*.

THERMAL CONNECTION

Select the **Connection type** from the list: **Conductive thermal resistor** (default), **Thermal capacitor**, or **Lumped thermal system**.

Conductive Thermal Resistor

It models heat loss by conduction in a domain by a thermal resistance R (SI unit: K/W) between the two connectors. This resistance 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 Thermal Connection](#) for more details on the underlying theory.

- When **Specify** is **Thermal resistance** (default), 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** checkbox 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** checkbox 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** checkbox is selected in the **Thermal Connection** 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 **Thermal Connection** 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 between the connectors 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 is used. If the material properties are temperature-dependent, they are evaluated at the average temperature of the connectors.



Equivalent Thermal Conductivity Correlations

Optically Thick Participating Medium

This section is available when the **Optically thick participating medium** checkbox is selected in the **Thermal Connection** 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

$$k_R = \frac{16n_r^2\sigma(T_{avg})^3}{3\beta_R}$$

where n_r is the refractive index (dimensionless), σ is the Stefan-Boltzmann constant (SI unit: $W/(m^2 \cdot K^4)$), T_{avg} is the average temperature of the connector 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 **Thermal Connection** section is used. If the material properties are temperature-dependent, they are evaluated at the average temperature of the connector.

Thermal Capacitor

It models heat storage in a domain with a thermal capacitance C (SI unit: J/K) between the two connectors.

For a steady-state problem the temperature difference does not change with time and the imposed heat flux simplifies to zero.

See [Theory for Thermal Connection](#) for more details on the underlying theory.

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 pressure**, 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 pressure**, C_p . Select any material from the **Material** list to define **C_p From material**. For **User defined** enter a value or expression.

Lumped Thermal System

It models a thermal network defined by a **Lumped Thermal System** interface between the two connectors.

The average temperature on the selected boundaries for each connector 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 of each connector.

For each connector, 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. The initial temperature of the connector is also taken from the selected **External Terminal** feature for a transient simulation.

When the **Isothermal terminal** checkbox 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 checkbox to apply the constraint on the space-dependent temperature instead, which corresponds to a stronger constraint.



When only one **External Terminal** feature is selected in **Connector 1** or **Connector 2**, the behavior is equivalent to the **Lumped System Connector, Interface** feature.



[Theory for Thermal Connection](#)



LOCATION IN USER INTERFACE

Context Menus

Heat Transfer in Shells > Thermal Connection, Interface-Interface

Heat Transfer in Films > Thermal Connection, Interface-Interface

Heat Transfer in Fractures > Thermal Connection, Interface-Interface

Ribbon

Physics tab with **Heat Transfer in Shells** or any version of the Heat Transfer in Thin Structures interface selected:

Global > Thermal Connection, Interface-Interface

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** checkbox 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** checkbox 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** checkbox 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** checkbox 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

$$k_R = \frac{16n_r^2\sigma T^3}{3\beta_R}$$

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. 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, $T_{\text{ave}} = 0.5 \cdot (T_{p1} + T_{p2})$.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, 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 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**



Lumped Composite Thermal Barrier: Application Library path
**Heat_Transfer_Module/Tutorials_Thin_Structure/
lumped_composite_thermal_barrier**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Conductive Thermal Resistor

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Conductive Thermal Resistor

Convective Thermal Resistor

This feature models a two-port passive component of the thermal system. It connects two nodes, by creating a difference in the temperatures of its two connecting ports. It models heat loss by convection in a fluid of temperature T_{p2} over a solid wall of temperature T_{p1} .

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the following relation between the heat rate P and the temperature difference ΔT :

$$P = -\frac{\Delta T}{R_{\text{conv}}}$$

where R_{conv} (SI unit: K/W) is the thermal resistance, which is defined from the surface **Area** A and the **Heat transfer coefficient** h as

$$R_{\text{conv}} = \frac{1}{hA}$$

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

By default the **Temperature** is **User defined** and set to the port p1 temperature, $T_{\text{var}} = T_{p1}$.

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**, $\phi_{w,ext}$, 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 [Convective Heat Transfer Correlations](#).
- For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in [Ref. 21](#).

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,init}$, and the **Initial temperature at node 2**, $T_{2,init}$, to be used at initialization, in particular to evaluate the heat transfer coefficient.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature at node 1**
- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Convective Thermal Resistor

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Convective Thermal Resistor

External Terminal

Use this feature to connect a node (one-port component) of the thermal system to a distributed finite element model. It prescribes the temperature T_{ext} provided by a [Lumped System Connector](#) boundary feature of a Heat Transfer interface, a [Lumped System Connector, Interface \(Heat Transfer in Shells Interface\)](#) boundary interface feature of a Heat Transfer in Shells interface, or a [Lumped System Connector \(Heat Transfer in Shells Interface\)](#) edge interface feature of a Heat Transfer in Shells interface.

The **Lumped System Connector** feature uses the heat rate defined by the **External Terminal** feature on the selected node to set a heat flux on the corresponding boundary, interface, or edge in the distributed finite element model.

NODE CONNECTIONS

Set the **Node name** for the external terminal.

EXTERNAL CONNECTION

This section displays an information message about the connection status of the node:

- If the **External Terminal** feature is selected in the **Source** list of a **Thermal Connector** feature, the message **Current status: connected** is displayed.

- Else, the message **Current status: not connected** is displayed. The default value 293.15 K is used for T_{ext} .

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature**



Lumped Composite Thermal Barrier: Application Library path
**Heat_Transfer_Module/Tutorials,_Thin_Structure/
 lumped_composite_thermal_barrier**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Nodes > External Terminal

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_l **From material**. Alternatively, choose among **Water** and **Mercury** for a predefined material. For **User defined** enter a value or expression.

Select any material from the **Material** list to define the **Thermal conductivity, immobile** k_w **From material**. Alternatively, choose among **Copper** and **Aluminum** for a predefined material. For **User defined** enter a value or expression.

Then, set a value or expression for the **Porosity**, ϵ , and select an option from the **Structure** list to calculate the effective conductivity. See [Theory for the Heat Pipe Component](#) for details.

Alternatively, set **Input quantity** to **User defined** to set directly the **Effective thermal conductivity of the wick**, k_{eff} .

Operating Maximum Power

Finally, select the **Specify operating maximum power** checkbox 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

Select an option from the **Specify** list: **Heat rate** (default), **Convective heat rate**, **Nucleate boiling heat rate**, or **Joule heating**.

Heat Rate

Enter a value or expression for P_0 to prescribe it as the heat rate at the selected node of the thermal system.

Convective Heat Rate

The heat rate P is defined in function of the difference between the node temperature T and a specified external temperature, T_{ext} , by mean of a convective thermal resistance, R_{conv} :

$$P = \frac{T_{\text{ext}} - T}{R_{\text{conv}}}$$

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 [Convective Heat Transfer Correlations](#).
- For a thorough introduction about how to calculate heat transfer coefficients, see Incropera and DeWitt in [Ref. 21](#).

Nucleate Boiling Heat Rate

The heat rate P is defined as follows:

$$P = Aq_0$$

where A is surface area, and q_0 is computed with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See [Nucleate Pool Boiling Correlation](#) for details about the correlation. Set the **Area** A of the surface of the nucleate boiling heat flux, and select materials from the **Fluid** and **Surface** lists. When a predefined material (**Water**, **Benzene**, **n-Pentane**, or **Ethanol**) is used for the fluid, no additional parameters are required, because the liquid and vapor properties are predefined (at saturation temperature) and the Rohsenow's correlation parameters C_{sf} and s are available for different surface materials. Else, see [Nucleate Boiling Heat Flux](#) ([Heat Flux](#) feature) for details about the settings.

Joule Heating

The heat rate P is defined as follows to model resistive heating at the node:

$$P = R_e I^2$$

where R_e is the electric resistance and I is the electric current.

Set values or expressions for the **Electric current**, I , and the **Electric resistance**, R_e .

INITIAL VALUES

Set a user-defined value or expression for the **Initial temperature**, T_{init} , to be used at initialization, in particular to evaluate the heat transfer coefficient.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Nodes > Heat Rate

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Nodes > Heat Rate

Heat Rate Source

This feature models a two-port source component of the thermal system. It connects two nodes, by creating a difference in the heat rates of its two connecting ports. It models a heat source or sink in a domain.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the heat rate P .

IDENTIFIER

Enter a **Component name** for the heat rate source. The prefix is P.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the heat rate source.

COMPONENT PARAMETERS

Set a value or expression for the heat rate P_{src} .

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$.

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 Rate Source

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Heat Rate Source

Nucleate Boiling Heat Rate

This feature models a two-port source component of the thermal system. It connects two nodes, by creating a difference in the heat rates of its two connecting ports. It models a heat sink due to nucleate boiling.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the heat rate P_{src} .

IDENTIFIER

Enter a **Component name** for the nucleate boiling heat rate. The prefix is NBP.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the nucleate boiling heat rate.

COMPONENT PARAMETERS

The heat rate P_{src} is defined as follows:

$$P_{\text{src}} = Aq_0$$

where A is surface area, and q_0 is computed with the Rohsenow's correlation, that evaluates the heat flux due to nucleate boiling on a surface immersed in a liquid pool. See [Nucleate Pool Boiling Correlation](#) for details about the correlation. Set the **Area** A of the surface of the nucleate boiling heat flux, and select materials from the **Fluid** and **Surface** lists. When a predefined material (**Water**, **Benzene**, **n-Pentane**, or **Ethanol**) is used for the fluid, no additional parameters are required, because the liquid and vapor properties are predefined (at saturation temperature) and the Rohsenow's correlation parameters C_{sf} and s are available for different surface materials. Else, see [Nucleate Boiling Heat Flux](#) ([Heat Flux](#) feature) for details about the settings.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,init}$, and the **Initial temperature at node 2**, $T_{2,init}$.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature at node 1**
- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Nucleate Boiling Heat Rate

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Nucleate Boiling Heat Rate

Radiative Heat Rate

This feature models a node (one-port component) of the thermal system, at which a heat rate due to surface-to-ambient radiation is prescribed.

NODE CONNECTIONS

Set the **Node name** for the radiative heat rate node.

NODE PARAMETERS

The heat rate P due to radiation on a diffuse-gray surface from the ambient surrounding is:

$$P = \varepsilon \sigma A (T_{\text{amb}}^4 - T^4)$$

where σ is the Stefan-Boltzmann constant, ε is the emissivity of the surface, A is the area of the surface, T_{amb} is the ambient temperature, and T is the surface temperature.

In the thermal system, the corresponding heat rate applied on the selected node can be expressed in function of the difference between the ambient temperature, T_{amb} and the node temperature, T , by mean of the radiative thermal resistance, R_{rad} :

$$P = \frac{T_{\text{amb}} - T}{R_{\text{rad}}}$$

where

$$R_{\text{rad}} = \frac{1}{\varepsilon \sigma A (T_{\text{amb}}^2 + T^2)(T_{\text{amb}} + T)}$$

First, set the surface **Area** A . Then, select any material from the **Material** list to define the **Surface emissivity** ε **From material**. For **User defined** enter a value or expression.

Finally, set the **Ambient temperature**, T_{amb} . For **User defined**, enter a value or expression. Else, select an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**.

INITIAL VALUES

Set a user-defined value or expression for the **Initial temperature**, T_{init} , to be used at initialization, in particular to evaluate the surface emissivity.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Nodes > Radiative Heat Rate

Ribbon

Physics tab with **Lumped Thermal System** selected:

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**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$, to be used at initialization, in particular to evaluate the surface emissivities.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature at node 1**
- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Radiative Thermal Resistor

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Radiative Thermal Resistor

Subsystem Definition

This feature is used to define subsystems, which can be inserted as components into the main system using a [Subsystem Instance](#) node. Create the subsystem by adding subnodes to the **Subsystem Definition** node.

NODE CONNECTIONS

Define the number of connecting nodes by adding more nodes or deleting existing nodes and set the **Node names** at which the subsystem connects to the main system or to other subsystems. The order in which the **Node names** are defined is the order in which they are referenced by a **Subsystem Instance** node.

The components constituting the subsystem should be connected only to the subsystem's nodes and to themselves.

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Subsystems > Subsystem Definition

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Subsystems > Subsystem Definition

Subsystem Instance

This feature represents an instance of a subsystem defined by a [Subsystem Definition](#) feature.

IDENTIFIER

Enter a **Component name** for the subsystem instance. The prefix is X.

NODE CONNECTIONS

Select the **Name of subsystem link** from the list of defined subsystems in the model and set the **Node names** at which the subsystem instance connects to the main system or to another subsystem.

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Subsystems > Subsystem Instance

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Subsystems > Subsystem Instance

Temperature

This feature models a node (one-port component) of the thermal system, used to prescribe the temperature.

NODE CONNECTIONS

Set the **Node name** for the temperature node.

NODE PARAMETERS

Set a value or expression for the **Temperature**, T_0 .

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Nodes > Temperature

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Nodes > Temperature

Thermal Capacitor

This feature models a two-port passive component of the thermal system. It connects two nodes, by creating a difference in the temperatures of its two connecting ports. It models heat storage in a domain.

It adds equations for the heat rates P_{p1} and P_{p2} and the temperatures T_{p1} and T_{p2} at the connecting ports p1 and p2 of the component, and defines the following relation between the heat rate P and the temperature difference ΔT :

$$P = -C \frac{\partial \Delta T}{\partial t} \quad (6-20)$$

where C (SI unit: J/K) is the thermal capacitance, defined as

$$C = V\rho C_p = mC_p$$

with V the volume (SI unit: m³), ρ the density (SI unit: kg/m³), C_p the heat capacity at constant pressure (SI unit: J/(kg·K)), and m the mass (SI unit: kg).

For a steady-state problem the temperature difference does not change with time and $P = 0$.

See [Theory for the Thermal Capacitor and Thermal Mass Components](#) for more details on the underlying theory.

IDENTIFIER

Enter a **Component name** for the thermal capacitor. The prefix is C.

NODE CONNECTIONS

Set the two **Node names** for the nodes connected by the thermal capacitor.

COMPONENT PARAMETERS

The thermal capacitance used in [Equation 6-20](#) should be set in this section.

Depending on the option selected in the **Specify** list, different settings are required:

- With **Thermal capacitance** (default), set directly a value or expression for C .
- With **Density and heat capacity**, set values or expressions for the **Volume**, V , the **Density**, ρ , and the **Heat capacity at constant pressure**, 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 pressure**, 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:

$$P = -C \frac{\partial T}{\partial t} \quad (6-21)$$

where C (SI unit: J/K) is the thermal capacitance, defined as

$$C = V\rho C_p = mC_p$$

with V the volume (SI unit: m³), ρ the density (SI unit: kg/m³), C_p the heat capacity at constant pressure (SI unit: J/(kg·K)), and m the mass (SI unit: kg).

This component is similar to the [Thermal Capacitor](#) two-port component, with the difference that the thermal mass has a virtual fixed temperature node.

For a steady-state problem the temperature does not change with time and $P = 0$.

See [Theory for the Thermal Capacitor and Thermal Mass Components](#) for more details on the underlying theory.

NODE CONNECTIONS

Set the **Node name** for the thermal mass node.

NODE PARAMETERS

The thermal capacitance used in [Equation 6-21](#) should be set in this section.

Depending on the option selected in the **Specify** list, different settings are required:

- With **Thermal capacitance** (default), set directly a value or expression for C .
- With **Density and heat capacity**, set values or expressions for the **Volume**, V , the **Density**, ρ , and the **Heat capacity at constant pressure**, 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 pressure**, 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:

$$P_{p1} = -SIT_{p1} + \frac{R_e I^2}{2} + \frac{\Delta T}{R} - C_1 \frac{\partial T_{p1}}{\partial t} \quad (6-22)$$

- p2 side:

$$P_{p2} = SIT_{p2} + \frac{R_e I^2}{2} - \frac{\Delta T}{R} - C_2 \frac{\partial T_{p2}}{\partial t} \quad (6-23)$$

where

- S (SI unit: V/K) is the Seebeck coefficient of the thermocouples
- I (SI unit: A) is the electric current operating in the module
- R_e (SI unit: Ω) is the electric resistance of the thermocouples
- ΔT (SI unit: K) is the temperature difference between the two sides of the module
- R (SI unit: K/W) is the thermal resistance of the thermocouples
- C_1 and C_2 (SI unit: J/K) are the thermal capacitances on each side of the module

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

When summing [Equation 6-22](#) and [Equation 6-23](#) to obtain the total heat rate through the thermoelectric module, the terms for heat dissipation by conduction cancel each other. Note that the material properties are evaluated at the average temperature of the two ports temperatures.

See [Theory for the Thermoelectric Module Component](#) for details.

For a thermoelectric cooler, the ports p1 and p2 correspond respectively to the cold and hot sides of the module, and the cold side is the side where the heat should be removed.

In addition, the total heat rate through the thermoelectric module may be expressed from performance parameters:

- p1 side:

$$P_{p1} = -\left(Q_{\max} - \Delta T \frac{Q_{\max}}{\Delta T_{\max}}\right) \quad (6-24)$$

- p2 side:

$$P_{p2} = Q_{\max} - \Delta T \frac{Q_{\max}}{\Delta T_{\max}} + R_e I^2 \quad (6-25)$$

where

- $Q_{\max}$ (SI unit: W) is the maximum removed heat on the cold side
- ΔT (SI unit: K) is the temperature difference between the two sides of the module
- $\Delta T_{\max}$ (SI unit: K) is the maximum temperature difference across the module
- R_e (SI unit: Ω) is the electric 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, $T_{ave} = 0.5 \cdot (T_{p1} + T_{p2})$, 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 6-24](#) and [Equation 6-25](#)) to characterize the module. Then, set values or expressions for the operating **Electric current**, I , and set the other parameters in the [Performance Parameters](#) section.
- Set **Specify** to **Performance parameters, general model** to apply user defined heat rates on each side of the module. Set values or expressions for the **Heat rate, node 1** and the **Heat rate, node 2**. In this formulation, the nodes 1 and 2 may refer to any side of the module, and the heat rate expressions may depend on the temperature difference and on the current intensity, among other variables. The temperature difference can be accessed through the `lts.TEM1.DT` variable, when the component name is TEM1, in a physics interface with tag `lts`.
- Set **Specify** to **Thermal and geometric properties** to use [Equation 6-22](#) and [Equation 6-23](#) to characterize the module. Set values or expressions for the operating **Electric current**, I , and the **Number of thermocouples**, N (composed each of one p-type and one n-type semiconductors) in the thermoelectric module. Then the material and geometric properties should be set in the [P-Type Semiconductor Parameters](#), [N-Type Semiconductor Parameters](#), and [Ceramic Plate Parameters](#) section that display underneath.

PERFORMANCE PARAMETERS

This section is available when **Specify** is set to **Performance parameters, TEC linearized model** in the **Component Parameters** section.

Set the **Maximum heat rate, cold side**, $Q_{\max}$, the **Maximum temperature difference**, $\Delta T_{\max}$, and the **Electric resistance**, R_e , used for the expression of the heat rates on each side, following [Equation 6-24](#) and [Equation 6-25](#).

P-TYPE SEMICONDUCTOR PARAMETERS

This section is available when **Specify** is set to **Thermal and geometric properties** in the **Component Parameters** section. The material and geometric properties of the p-type semiconductors should be set in this section.

First, set the **Seebeck coefficient**, S_p .

Electric and Thermal Resistances

Either specify directly the **Electric resistance**, $R_{e,p}$ and **Thermal resistance**, R_p or specify the **Electric conductivity**, $\sigma_{e,p}$ and **Thermal conductivity**, k_p to calculate the corresponding resistances.

Thermal Capacitance

Either specify directly the **Thermal capacitance**, C_p or specify the **Density**, ρ_p and **Heat capacity at constant pressure**, $C_{p,p}$ to calculate the corresponding thermal capacitance. For a steady-state problem, heat storage is null and these inputs are ignored.

Geometry Parameters

The **Section area**, A_p and **Length**, L_p of the p-type semiconductors are needed when either the resistances are expressed from the conductivities or the capacitance is expressed from the density and heat capacity at constant pressure.

Material

Select any material from the **Material** list to define some material properties **From material**. For **User defined** enter values or expressions for these properties.

N-TYPE SEMICONDUCTOR PARAMETERS

This section is available when **Specify** is set to **Thermal and geometric properties** in the **Component Parameters** section. The material and geometric properties of the n-type semiconductors should be set in this section. See [P-Type Semiconductor Parameters](#) section for details.

CERAMIC PLATE PARAMETERS

This section is available when **Specify** is set to **Thermal and geometric properties** in the **Component Parameters** section. The material and geometric properties of the ceramic plates should be set in this section.

Thermal Capacitance

Either specify directly the **Thermal capacitance**, C_c or specify the **Density**, ρ_c and **Heat capacity at constant pressure**, $C_{p,c}$ to calculate the corresponding thermal capacitance. For a steady-state problem, heat storage is null and these inputs are ignored.

Geometry Parameters

The **Section area**, A_c and **Thickness**, d_c of the ceramic plates are needed when the capacitance is expressed from the density and heat capacity at constant pressure.

Material

Select any material from the **Material** list to define some material properties **From material**. For **User defined** enter values or expressions for these properties.

INITIAL VALUES

Set user defined values or expressions for the **Initial temperature at node 1**, $T_{1,\text{init}}$, and the **Initial temperature at node 2**, $T_{2,\text{init}}$, to be used at initialization, in particular to evaluate the material properties, the performance parameters of the thermoelectric module, and the heat storage term.

RESULTS

Select appropriate options in the **Add the following to default results** in order to include the following global variables (space-independent) in the default plots:

- **Heat rate**
- **Temperature at node 1**
- **Temperature at node 2**

LOCATION IN USER INTERFACE

Context Menus

Lumped Thermal System > Thermoelectric Module

Ribbon

Physics tab with **Lumped Thermal System** selected:

Global > Thermoelectric Module

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](#)



For details about the features available in the **Moisture Transport in Solids** interface, see [Domain and Boundary Nodes for the Moisture Transport in Solids Interface](#) in the *Porous Media Flow Module User's Guide*.

Domain Features

The Moisture Transport interface has the following domain nodes available:

- Building Material
- Hygroscopic Porous Medium
- Initial Values
- Liquid Water (Hygroscopic Porous Medium)
- Liquid Water Source
- Moist Air (Hygroscopic Porous Medium)
- Moist Air (Moisture Transport Interface)
- Moisture Source
- Porous Matrix (Hygroscopic Porous Medium)
- Turbulent Mixing
- Vapor Source



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

$$\xi \frac{\partial \phi_w}{\partial t} + \nabla \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = G \quad (7-1)$$

$$\mathbf{g}_w = -\delta_p \nabla (\phi_w p_{sat}) \quad (7-2)$$

$$\mathbf{g}_{lc} = -\xi D_w \nabla \phi_w$$

$$\xi = \frac{\partial w(\phi_w)}{\partial \phi_w}$$

with the following material properties, fields, and source:

- ξ (SI unit: kg/m³) 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**, select **Isotropic**, **Diagonal**, or **Symmetric** based on the vapor permeability characteristics, and set values 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**, select **Isotropic**, **Diagonal**, or **Symmetric** based on the vapor permeability characteristics and set its value.
- **Vapor resistance factor** μ to define the vapor permeability δ_p as:

$$\delta_p = \mu^{-1}\delta$$

where δ (SI unit: s) is the vapor permeability of still air. The default **Vapor resistance factor** is taken **From material**. For **User defined**, set a value.

	The Moisture Transport in Building Materials Interface
	The Heat and Moisture Transport Interfaces
	Heat and Moisture
	Theory for Moisture Transport in Building Materials
	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.
	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/
	<i>Heat and Moisture Transport in a Semi-Infinite Wall</i> : Application Library path Heat_Transfer_Module/Verification_Examples/semi_infinite_wall

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Building Material

Moisture Transport in Building Materials > Building Material

Equilibrium Moisture Transport in Porous Media > Building Material

Nonequilibrium Moisture Transport in Porous Media > Building Material

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Domains > Building Material

Hygroscopic Porous Medium

Use this node to model moisture transport in a porous medium filled with two phases: liquid water and moist air. Phases can be either in equilibrium or not, depending on the **Equilibrium between liquid and gas phases** option. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

Equilibrium Formulation

With this assumption, 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.

In the equilibrium formulation, the evaporation and condensation source terms from the moist air and liquid water mass conservation equations cancel out. However, the **Hygroscopic Porous Medium** node stores the evaporation source term into the `mt.G_evap` variable, to be accounted for as a mass source in the fluid flow equations for the gas phase in a moisture flow model. This mass source is automatically handled by the [Moisture Flow](#) coupling node.

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.

Nonequilibrium Formulation

When liquid water and water vapor are not in equilibrium, the equations of mass conservation of water are solved separately for both phases. They are coupled through a transfer term which accounts for evaporation, and is proportional to the difference between the partial vapor pressure and the equilibrium vapor pressure. See [Theory for Moisture Transport in Porous Media](#) for details.

MODEL INPUT

This section has fields and values that are inputs to expressions that define material properties in the **Liquid Water**, **Moist Air**, and **Porous Matrix** subnodes.

The default **Temperature** T and **Absolute pressure** p_A are **User defined**. When additional physics interfaces are added to the model, the temperature and absolute pressure variables defined by these physics interfaces can also be selected from the list. For example, if a **Heat Transfer in Moist Porous Media** interface is added, you can select **Temperature (ht)** from the list. If a **Laminar Flow** interface is added, you can select **Absolute pressure (spf)** from the list.

If the node was added automatically after selecting the **Heat and Moisture Transport** predefined multiphysics interface, the temperature of the **Heat and Moisture** multiphysics node is used by default and the input field is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** ().

COORDINATE SYSTEM SELECTION

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate system**, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The **Hygroscopic Porous Medium** node and its subnodes inherit these coordinate system settings. In particular, the **Moist air velocity field** (in [Moist Air \(Hygroscopic Porous Medium\)](#) subnode), the **Permeability** (in [Porous Matrix \(Hygroscopic Porous Medium\)](#) subnode), and the **Capillary liquid flux density** (in [Hygroscopic Porous Medium](#) node) should be set according to the coordinate system selected in this section.

See [Coordinate Systems](#) in the *COMSOL Multiphysics Reference Manual* for more details.

MOISTURE TRANSPORT PROPERTIES

Select between the **Equilibrium formulation** and the **Nonequilibrium formulation** options in the **Equilibrium between liquid and gas phases** list.

Depending upon the selected option, further settings are required underneath: either the moisture storage function required by the single mass conservation equation, or the parameters used to couple the two mass conservation equations in each phase. See the [Equilibrium Formulation](#) and [Nonequilibrium Formulation](#) sections below for details.

It also changes the availability of some subnodes:

- The [Initial Values](#), [Insulation](#), [Symmetry](#), [Liquid Water Content](#), and [Liquid Water Flux](#) features are available under the [Liquid Water \(Hygroscopic Porous Medium\)](#) subnode only when **Equilibrium between liquid and gas phases** is set to **Nonequilibrium formulation**. These subnodes allow the definition of domain and boundary conditions specific to the liquid water saturation s_l .
- The [Initial Values](#), [Insulation](#), [Symmetry](#), [Vapor Content](#), [Vapor Flux](#), [Inflow](#), [Outflow](#), and [Open Boundary](#) features are available under the [Moist Air \(Hygroscopic Porous Medium\)](#) subnode only when **Equilibrium between liquid and gas phases** is set to **Nonequilibrium formulation**. These subnodes allow the definition of domain and boundary conditions specific to the relative humidity ϕ_w .

Equilibrium Formulation

When **Equilibrium formulation** is selected in the **Equilibrium between liquid and gas phases** list, this section defines the **Moisture storage function** $w(\phi_w)$. It characterizes the relationship between the amount of accumulated water and the relative humidity in the porous material. The default **Moisture storage function** is taken **From material**. For **User defined**, enter another value or expression.

When a **Porous Material** node is defined on the domains where the **Hygroscopic Porous Medium** feature is active, the **Moisture storage function** can be set in the **Homogenized Properties** section of the **Porous Material** node.

Nonequilibrium Formulation

When **Nonequilibrium formulation** is selected in the **Equilibrium between liquid and gas phases** list, this section defines the equilibrium vapor pressure $p_{v,eq}$ (SI unit: Pa) and the evaporation rate K_{evap} (SI unit: 1/s).

In the **Equilibrium vapor pressure** list, select **Water activity** to define the equilibrium vapor pressure from the saturation pressure $p_{v,sat}$ (SI unit: Pa) and the water activity a_w (dimensionless)

$$p_{v,eq} = a_w p_{v,sat}$$

Enter a value for the **Water activity** in the porous medium. Default value is 1.

Select **User defined** to enter an expression for $p_{v,eq}$. Default expression is `mt.fpsat(mt.T)`, which gives the saturation pressure at the prescribed temperature.

Enter a value for the **Evaporation rate**. Default value is 0 [1/s].

	See Porous Material in the <i>COMSOL Multiphysics Reference Manual</i> .
	The properties for liquid water transport are specified in the Liquid Water (Hygroscopic Porous Medium) subnode. The properties for moist air transport are specified in the Moist Air (Hygroscopic Porous Medium) subnode. The solid matrix porosity and permeability are specified in the Porous Matrix (Hygroscopic Porous Medium) subnode. When the Hygroscopic Porous Medium node is coupled to a Porous Medium node in a fluid flow interface through the Moisture Flow coupling, the porosity and moist air permeability are automatically shared and corrected for the computation of the Darcy velocity of moist air to account for the presence of liquid water.
	• The Equilibrium Moisture Transport in Porous Media Interface • The Nonequilibrium 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

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface 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**.

When this node is added as a subnode of a **Moist Air (Hygroscopic Porous Medium)** node, or a **Liquid Water (Hygroscopic Porous Medium)** node, it controls the initial **Relative humidity** $\phi_{w,0}$ or the initial **Liquid water saturation** $s_{l,0}$ respectively.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Initial Values

Moisture Transport in Building Materials > Initial Values

Equilibrium Moisture Transport in Porous Media > Initial Values

Nonequilibrium Moisture Transport in Porous Media > Initial Values

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Liquid Water (Hygroscopic Porous Medium)

This subnode defines the properties for the transport of the liquid water phase in a porous medium filled with two phases: liquid water and moist air. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

LIQUID WATER PROPERTIES

This section contains the settings for the definition of convective and capillary transport of liquid water.

Depending upon the selected option for **Equilibrium between liquid and gas phases** in the parent [Hygroscopic Porous Medium](#) node, different options are available for the capillary flux. If **Equilibrium formulation** is selected:

- When **Capillary model** is set to **Kelvin's law** (default), a Darcy's law based on the capillary pressure is used. The capillary pressure is derived from the relative humidity by using Kelvin's law, using the assumption that the liquid and gas phases are in equilibrium.
- When **Capillary model** is set to **Diffusion**, the capillary flux is derived from the gradient of the moisture content, and the **Moisture diffusivity** D_w .
- For **User defined**, set a value for the **Capillary liquid flux density**, $\mathbf{g}_{lc}$, to characterize the liquid transport as a function of the moisture content.

If **Nonequilibrium formulation** is selected:

- When **Capillary model** is set to **Diffusion** (default), the capillary flux is derived from the gradient of the moisture content, and the **Moisture diffusivity** D_w .
- When **Capillary model** is set to **Pressure**, the capillary flux is derived from the gradient of capillary pressure. In turn, the **Pressure model** gives the expression of the capillary pressure. Details are given below.
- For **User defined**, set a value for the **Capillary liquid flux density**, $\mathbf{g}_{lc}$, to characterize the liquid transport as a function of the moisture content.

Different options are available for the **Pressure model**:

- When **Pressure model** is set to **van Genuchten** (default), the capillary pressure is calculated from the **Entry capillary pressure** p_{ec} (SI unit: Pa), for which the default value is 10 [kPa], and two constitutive relation constants m_{vG} (dimensionless), for

which the default value is 0.3, and l_{vG} (dimensionless), for which the default value is 0.5. This model also gives the value of the relative liquid water permeability κ_{rl} (dimensionless) and of the relative moist air permeability κ_{rg} (dimensionless). The first one is used to calculate the capillary liquid flux, and the second one is used in the [Moisture Flow](#) coupling.

- When **Pressure model** is set to **Brooks and Corey**, the capillary pressure is calculated from the **Entry capillary pressure** p_{ec} (SI unit: Pa), for which the default value is 10 [kPa], and the pore size distribution index λ_p (dimensionless), for which the default value is 2. This model also gives the value of the relative liquid water permeability κ_{rl} and of the relative moist air permeability κ_{rg} . The first one is used to calculate the capillary liquid flux, and the second one is used in the [Moisture Flow](#) coupling.
- For **User defined**, set an expression for the **Capillary pressure**, p_c , to characterize the liquid transport as a function of the liquid saturation. The default expression is $10 \text{ [kPa]} * (1 - m_t.s1)$.

When a **Porous Material** node is defined on the domains where the **Hygroscopic Porous Medium** feature is active, the **Moisture diffusivity** can be set in the **Homogenized Properties** section of the **Porous Material** node.

The convective flux of liquid water is always computed with Darcy's law. When not calculated by the capillary model, set the **Relative liquid water permeability**, κ_{rl} , that multiplies the porous medium permeability in the Darcy's law to account for the saturation of the liquid phase. The relative liquid water permeability κ_{rl} is a function of the liquid water saturation s_l . In simplest models it can be set as $\kappa_{rl} = s_l$.



The liquid saturation, s_l , is defined from the moisture storage function specified in the [Hygroscopic Porous Medium](#) parent node when **Equilibrium between liquid and gas phases** is set to **Equilibrium formulation**, and is a dependent variable when it is set to **Nonequilibrium formulation**.

	The liquid water properties are based on the temperature set in the corresponding model input in the Hygroscopic Porous Medium parent node. The properties for phase distribution of moisture content in the porous material are specified in the Hygroscopic Porous Medium parent node. The properties for moist air transport are specified in the Moist Air (Hygroscopic Porous Medium) subnode. The solid matrix porosity and permeability are specified in the Porous Matrix (Hygroscopic Porous Medium) subnode. When the parent Hygroscopic Porous Medium node is coupled to a Porous Medium node in a fluid flow interface through the Moisture Flow coupling, the porosity and moist air permeability are automatically shared and corrected for the computation of the Darcy velocity of moist air to account for the presence of liquid water.
	The Equilibrium Moisture Transport in Porous Media Interface The Nonequilibrium Moisture Transport in Porous Media Interface The Heat and Moisture Transport Interfaces Heat and Moisture Theory for Moisture Transport in Porous Media
	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Hygroscopic Porous Medium > Liquid Water

Moisture Transport in Building Materials > Hygroscopic Porous Medium > Liquid Water

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water

Liquid Water Source

This node describes moisture generation in the form of liquid water within the hygroscopic porous medium 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 **Liquid Water Source** node adds a source term G_1 to the right-hand side of the liquid water transport equation:

$$G_1 = G_{1,0}$$

LIQUID WATER SOURCE

Enter a value or expression for the **Liquid Water Source** $G_{1,0}$ per unit volume.



This feature is intended to model additional external moisture sources or sinks, such as those that would result from a chemical reaction like combustion where water is a reaction product, or from water phase change with a phase that is not modeled. In particular, this feature can be used to account for the additional liquid water that would enter a hygroscopic porous medium due to the melting of ice, since the solid water composing the ice cannot be modeled with **Moisture Transport** interfaces. But it should not be used to model condensation in a hygroscopic porous medium since liquid water is already modeled. The latter phase change automatically happens under the favorable conditions.



When using a **Heat and Moisture** coupling, sources added with the **Liquid Water Source** feature are not accounted for in the heat transfer equations. If the generation or consumption of liquid water is associated to a thermodynamic process that generates or consumes heat, it should be manually added in the **Heat Transfer** interface.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Hygroscopic Porous Medium > Liquid Water > Liquid Water Source
Moisture Transport in Building Materials > Hygroscopic Porous Medium > Liquid Water > Liquid Water Source

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water > Liquid Water Source
Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water > Liquid Water Source

Ribbon

Physics tab with **Hygroscopic Porous Medium > Liquid Water** selected in the model tree:

Attributes > Liquid Water Source

Moist Air (Hygroscopic Porous Medium)

This subnode defines the properties for the transport of the moist air phase in a porous medium filled with two phases: liquid water and moist air. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

MOIST AIR PROPERTIES

This section contains settings for the definition of convective and diffusive vapor transport in the moist air phase.

Set the **Diffusion coefficient** D (SI unit: m^2/s) for the binary diffusion of vapor in air. The default value $2.6 \cdot 10^{-5} \text{ m}^2/\text{s}$ is valid at $T = 298 \text{ K}$ and $p_A = 1 \text{ atm}$. You may set a temperature or pressure-dependent diffusion coefficient through the use of the variables mt.T and mt.pA .

Specify the **Effective diffusivity model** used to account for the porosity and the tortuosity of the porous medium in the diffusion coefficient, D_{eff} (SI unit: m^2/s). The available options are **Millington and Quirk model** (the default), **Bruggeman model**, **Tortuosity model**, and **No correction**.

For the **Tortuosity model** option, enter a value for the **Tortuosity factor**, τ (dimensionless). The default is 1.

Then, set the **Moist air velocity field**, $\mathbf{u}_g$, that should be interpreted as the Darcy velocity, that is, the volume flow rate per unit cross sectional area. For **User defined** enter values or expressions for the components based on space dimensions. Or select an existing velocity field in the component (for example, **Velocity field (br)** from a **Brinkman Equations** interface).

When the parent [Hygroscopic Porous Medium](#) node is coupled to a [Porous Medium](#) node in a fluid flow interface through the [Moisture Flow](#) coupling, an additional

property, the relative moist air permeability κ_{rg} (dimensionless) can be set to account for the presence of liquid water in the computation of the Darcy velocity of moist air. The default value is $1\text{-m}\cdot\text{s}$. Depending on the chosen capillary model, this setting can be automatically calculated, and thus is not shown. See [Liquid Water \(Hygroscopic Porous Medium\)](#) for more details.

?	When setting temperature or pressure-dependent properties, the corresponding model inputs are specified in the Hygroscopic Porous Medium parent node. The properties for phase distribution of moisture content in the porous material are specified in the Hygroscopic Porous Medium parent node. The properties for liquid water transport are specified in the Liquid Water (Hygroscopic Porous Medium) subnode. The solid matrix porosity and permeability are specified in the Porous Matrix (Hygroscopic Porous Medium) subnode.
?	The Equilibrium Moisture Transport in Porous Media Interface The Nonequilibrium 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 > Moist Air**
- Moisture Transport in Building Materials > Hygroscopic Porous Medium > Moist Air**
- Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air**
- Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air**

Moist Air (Moisture Transport Interface)

Use this node to model moisture transfer in moist air through vapor diffusion and convection.

For small vapor concentration conditions, the density of moist air is supposed not to change with variations of moisture content, and a diluted species formulation is used. 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.

In conservative form, the following equation is solved instead of [Equation 7-3](#):

$$\frac{\partial(M_v c_v)}{\partial t} + \nabla \cdot (M_v c_v \mathbf{u}) + \nabla \cdot \mathbf{g}_w = G \quad (7-4)$$

For larger vapor concentration conditions, the variation of moist air density are modeled, and a concentrated species formulation is used. In this case, an additional term accounting for compressibility effects is added to the equation. Also, the vapor diffusive flux is written in terms of vapor mass fraction ω_v :

$$M_v \frac{\partial c_v}{\partial t} + M_v \mathbf{u} \cdot \nabla c_v + \nabla \cdot \mathbf{g}_w + M_v c_v \nabla \cdot \mathbf{u} = G \quad (7-5)$$

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

$$\omega_v = \frac{M_v c_v}{\rho_g}$$

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

where ρ_g is the moist air density (SI unit: kg/m³).

In conservative form, the following equation is solved instead of [Equation 7-5](#):

$$\frac{\partial(M_v c_v)}{\partial t} + \nabla \cdot (M_v c_v \mathbf{u}) + \nabla \cdot \mathbf{g}_w = G \quad (7-6)$$

Note that this formula is the same as for the diluted species formulation, but the expression of the vapor diffusive flux remains different.

For a steady-state problem, the relative humidity does not change with time and the first term in [Equation 7-3](#), [Equation 7-4](#), [Equation 7-5](#), and [Equation 7-6](#) disappears.

MODEL INPUT

This section has fields and values that are inputs to expressions used in [Equation 7-3](#), [Equation 7-4](#), [Equation 7-5](#), and [Equation 7-6](#). 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 $2.6\text{e-}5\text{ m}^2/\text{s}$ 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 $mt.pA$.

	The Moisture Transport in Air Interface The Heat and Moisture Transport Interfaces Heat and Moisture Theory for Moisture Transport in Air
	See Turbulent Mixing to account for eddy diffusivity in moisture convection by turbulent flows.
	For a detailed overview of the functionality available in each product, visit https://www.comsol.com/products/specifications/
	<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

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Moist Air

Moisture Transport in Building Materials > Moist Air

Equilibrium Moisture Transport in Porous Media > Moist Air

Nonequilibrium Moisture Transport in Porous Media > Moist Air

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface 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.



When added on a [Hygroscopic Porous Medium](#) domain with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, the moisture source is distributed between the liquid and vapor phases proportionally to the amount of liquid water and water vapor in the pores. The total moisture source is stored in the variable `mt.Gtot`, while the vapor and liquid water sources are stored in `mt.G_v` and `mt.G_l` respectively. See [Equilibrium Moisture Transport](#) for details.



This feature is intended to model additional external moisture sources or sinks, such as those that would result from a chemical reaction like combustion where water is a reaction product, or from water phase change with a phase that is not modeled. In particular, this feature can be used to account for the additional vapor that would enter moist air due to the sublimation of snow flakes, since the solid water composing the snow flakes cannot be modeled with **Moisture Transport** interfaces. But it should not be used to model evaporation in a hygroscopic porous medium since liquid water is already modeled. The latter phase change automatically happens under the favorable conditions.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Moisture Source

Moisture Transport in Building Materials > Moisture Source

Equilibrium Moisture Transport in Porous Media > Moisture Source
Nonequilibrium Moisture Transport in Porous Media > Moisture Source

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Domains > interface > Moisture Source

Porous Matrix (Hygroscopic Porous Medium)

This subnode defines the properties of a porous medium filled with two phases: liquid water and moist air. The moisture transport occurs through vapor diffusion and convection in moist air, and convection and capillary flow in the liquid phase.

POROUS MATRIX PROPERTIES

First, set the **Porosity**, ε_p (a dimensionless number between 0 and 1), that uses by default the value **From material**. For **User defined** the default is 0.

Then, set the **Permeability**, κ (SI unit: m^2), to specify the capacity of the porous material to transmit flow. For **User defined** select **Isotropic** to define a scalar value or **Diagonal**, **Symmetric**, or **Full** to define a tensor value and enter another value or expression in the field or matrix.

When a **Porous Material** node is defined on the domains where the **Hygroscopic Porous Medium** feature is active, the **Porosity** and the **Permeability** can be set in the **Homogenized Properties** section of the **Porous Material** node.

	When setting temperature or pressure-dependent properties, the corresponding model inputs are specified in the Hygroscopic Porous Medium parent node. The properties for phase distribution of moisture content in the porous material are specified in the Hygroscopic Porous Medium parent node. The properties for liquid water transport are specified in the Liquid Water (Hygroscopic Porous Medium) subnode. The properties for moist air transport are specified in the Moist Air (Hygroscopic Porous Medium) subnode. When the parent Hygroscopic Porous Medium node is coupled to a Porous Medium node in a fluid flow interface through the Moisture Flow coupling, the porosity and moist air permeability are automatically shared and corrected for the computation of the Darcy velocity of moist air to account for the presence of liquid water.
	The Equilibrium Moisture Transport in Porous Media Interface The Nonequilibrium 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 > Porous Matrix
- Moisture Transport in Building Materials > Hygroscopic Porous Medium > Porous Matrix

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Porous Matrix
Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Porous Matrix

Turbulent Mixing

This subnode should be used to account for the turbulent mixing caused by the eddy diffusivity in moisture convection, for example when the specified velocity field corresponds to a RANS solution.

The **Turbulent Mixing** node adds the following contribution to the diffusion coefficient D of [Equation 7-3](#), [Equation 7-4](#), [Equation 7-5](#), and [Equation 7-6](#) 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
Equilibrium Moisture Transport in Porous Media > Moist Air > Turbulent Mixing
Nonequilibrium Moisture Transport in Porous Media > Moist Air > Turbulent Mixing

Ribbon

Physics tab with **Moist Air** selected in the model tree:

Domains > Turbulent Mixing

Vapor Source

This node describes moisture generation in the form of water vapor within the hygroscopic porous medium 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 **Vapor Source** node adds a source term G_v to the right-hand side of the liquid water transport equation:

$$G_v = G_{v,0}$$

VAPOR SOURCE

Enter a value or expression for the **Vapor Source** $G_{v,0}$ per unit volume.

	This feature is intended to model additional external moisture sources or sinks, such as those that would result from a chemical reaction like combustion where water is a reaction product, or from water phase change with a phase that is not modeled. In particular, this feature can be used to account for the additional vapor that would enter a hygroscopic porous medium due to the sublimation of snow flakes, since the solid water composing the snow flakes cannot be modeled with Moisture Transport interfaces. But it should not be used to model evaporation in a hygroscopic porous medium since liquid water is already modeled. The latter phase change automatically happens under the favorable conditions.
	Sources added with the Vapor Source feature are not automatically accounted for in coupled interfaces. That is, when using a Heat and Moisture coupling, if the generation or consumption of vapor is associated to a thermodynamic process that generates or consumes heat, it should be manually added in the Heat Transfer interface. In the same way, when using a Moisture Flow coupling, the vapor source is not added to the continuity equation.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Hygroscopic Porous Medium > Moist Air > Vapor Source

Moisture Transport in Building Materials > Hygroscopic Porous Medium > Moist Air > Vapor Source

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Vapor Source

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Vapor Source

Ribbon

Physics tab with **Hygroscopic Porous Medium > Moist Air** selected in the model tree:

Attributes > Vapor Source

Boundary Features

The Moisture Transport interface has the following boundary nodes available:

- Continuity
- Free-Porous Interface
- Inflow
- Insulation
- Liquid Water Content
- Liquid Water Flux
- Moist Surface
- Moisture Content
- Moisture Flux
- Nonequilibrium Boundary
- Open Boundary
- Outflow
- Periodic Condition
- Porous Interface
- Porous-Porous Interface
- Symmetry
- Thin Moisture Barrier
- Vapor Content
- Vapor Flux
- Wet Surface



For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries into account and automatically adds an **Axial Symmetry** node that is valid on the axial symmetry boundaries only.



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

Continuity

This node can be added to pairs. It prescribes that the moisture content is continuous across the pair. **Continuity** is only suitable for pairs where the boundaries match. Depending on the domain conditions active in the adjacent domains, the constraint is applied either on relative humidity, vapor concentration or vapor mass fraction on boundary pairs. For continuous temperature and pressure conditions, all these constraints are equivalent.

PAIR SELECTION

Choose the pair on which to apply this condition. A pair has to be created first. See [Identity, Contact, and Sector Symmetry Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Nitsche constraints** (default), **Pointwise constraints** or **Weak constraints**. The **Nitsche constraints** option ensures continuity of the diffusive flux. This option may work better than the **Pointwise constraints** or **Weak constraints** when the meshes of the source and destination boundaries do not coincide. Alternatively, select the **Pointwise constraints** option. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings

ADVANCED

Select the **Disconnect pair** checkbox to disable the continuity condition.

LOCATION IN USER INTERFACE

Context Menus

- Moisture Transport in Air > Pairs > Continuity**
- Moisture Transport in Building Materials > Pairs > Continuity**
- Equilibrium Moisture Transport in Porous Media > Pairs > Continuity**
- Nonequilibrium Moisture Transport in Porous Media > Pairs > Continuity**

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Pairs > interface > Continuity

This node is active on a restriction of the parent [Porous Interface](#) feature where boundaries are adjacent to one porous medium feature, and one [Moist Air \(Moisture Transport Interface\)](#) feature. Here, porous medium features refer to the [Building Material](#) and the [Hygroscopic Porous Medium](#) features. Use this node to manage the fluxes continuity at the interface between a porous medium domain and a moist air domain.

FREE-POROUS INTERFACE

When **Phase change at the interface** is set to **No evaporation/condensation**, this condition imposes the continuity of the relative humidity ϕ_w , and the continuity of the total vapor flux. The total vapor flux is the sum of the vapor diffusive flux and the vapor convective flux in the **Moist Air** and the **Hygroscopic Porous Medium** features, while it is equal to the vapor diffusive flux only in the **Building Material** features, since there is no convection.

On interfaces between **Moist Air** and **Building Material** features:

$$\phi_w|_{MA} = \phi_w|_{BM}$$
$$(\mathbf{g}_w + M_v c_v \mathbf{u})|_{MA} \cdot \mathbf{n} = \mathbf{g}_w|_{BM} \cdot \mathbf{n}$$

On interfaces between **Moist Air** and **Hygroscopic Porous Medium** features:

$$\phi_w|_{MA} = \phi_w|_{HPM}$$
$$(\mathbf{g}_w + M_v c_v \mathbf{u})|_{MA} \cdot \mathbf{n} = (\mathbf{g}_w + M_v c_v \mathbf{u})|_{HPM} \cdot \mathbf{n}$$

Alternatively, you can choose the **Allow evaporation/condensation** option to impose the continuity of the relative humidity ϕ , and the continuity of the total moisture flux. The total moisture flux is the sum of the vapor diffusive flux and vapor convection flux in the **Moist Air** features since there is no liquid phase, the sum of the vapor diffusive flux and the capillary flux in the **Building Material** features, and the sum of the vapor diffusive flux, the capillary flux, and the liquid water and vapor convective fluxes in the **Hygroscopic Porous Medium** features.

On interfaces between **Moist Air** and **Building Material** features:

$$\phi_w|_{MA} = \phi_w|_{BM}$$

$$(\mathbf{g}_w + M_v c_v \mathbf{u})|_{\text{MA}} \cdot \mathbf{n} = (\mathbf{g}_w + \mathbf{g}_{lc})|_{\text{BM}} \cdot \mathbf{n}$$

On interfaces between **Moist Air** and **Hygroscopic Porous Medium** features:

$$\phi_w|_{\text{MA}} = \phi_w|_{\text{HPM}}$$

$$(\mathbf{g}_w + M_v c_v \mathbf{u})|_{\text{MA}} \cdot \mathbf{n} = (\mathbf{g}_w + M_v c_v \mathbf{u} + \mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_{\text{HPM}} \cdot \mathbf{n}$$

With this option phase change can occur, for example, the liquid flux on the side of the porous medium can become a vapor flux on the side of moist air. Use the **Contributes to heat flux** checkbox to account for it in the evaporation flux g_{evap} . Note that this flux is not directly used in the **Moisture Transport** interfaces, but in the **Heat and Moisture** coupling, where it is multiplied by the latent heat to be accounted for as a heat flux.

On interfaces between **Moist Air** and **Building Material** features:

$$g_{\text{evap}} = \mathbf{g}_{lc}|_{\text{BM}} \cdot \mathbf{n}$$

On interfaces between **Moist Air** and **Hygroscopic Porous Medium** features:

$$g_{\text{evap}} = (\mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_{\text{HPM}} \cdot \mathbf{n}$$

The associated heat flux accounted for in the **Heat and Moisture** coupling is:

$$q_{\text{evap}} = -L_v g_{\text{evap}}$$

Inflow

Use this node to model inflow of water vapor through a virtual domain, at inlet boundaries of moist air and hygroscopic porous medium domains. The upstream moisture conditions (either vapor concentration c_{ustr} , or vapor mass fraction ω_{ustr} , or relative humidity $\phi_{w,\text{ustr}}$ and temperature T_{ustr}) are known at the outer boundary of the virtual domain. The inflow condition is often similar to a moisture content condition. However using this condition the upstream vapor concentration is not strictly enforced at the inlet. The higher the flow rate, the smallest difference between the inlet and upstream moisture content. In case of moisture sources or moisture constraints close to the inlet boundary, the inflow condition induces a moisture profile that approaches to the moisture profile that would be obtained by computing the solution in the virtual domain upstream the inlet.

If both inflow and outflow conditions are expected at some boundaries, use the [Open Boundary](#) feature instead.

This boundary condition estimates the moisture flux through the inlet boundary from a Danckwerts condition on the vapor concentration:

$$-\mathbf{n} \cdot \mathbf{g}_w = M_v(c_v - c_{ustr})\mathbf{u} \cdot \mathbf{n}$$

Note that on the exterior boundaries of a porous medium, the moisture flux contains both the vapor diffusive flux and the liquid capillary flux:

$$-\mathbf{n} \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = M_v(c_v - c_{ustr})\mathbf{u} \cdot \mathbf{n}$$

Both the upstream vapor concentration c_{ustr} , and vapor mass fraction ω_{ustr} are defined on this boundary. Their expressions depend on the chosen inputs.

This node can be applied on the exterior boundaries of a computational domain in which either a [Moist Air \(Moisture Transport Interface\)](#) node, a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, or a [Moist Air \(Hygroscopic Porous Medium\)](#) subnode when it belongs to a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation** is active. In the latter case, this node only constrains the vapor flux.

UPSTREAM PROPERTIES

When **Specify** is set to **Relative humidity** in the **Specify** list, set both the **Upstream temperature** T_{ustr} and the **Upstream relative humidity** $\phi_{w,ustr}$ from which the upstream vapor concentration and the vapor mass fraction can be defined:

$$c_{ustr} = \phi_{w,ustr}c_{sat}(T_{ustr})$$

$$\omega_{ustr} = \frac{M_v c_{ustr}}{M_v c_{ustr} + M_a \left(\frac{p_A}{RT} - c_{ustr} \right)}$$

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} . The vapor mass fraction is then defined as

$$\omega_{\text{ustr}} = \frac{M_v c_{\text{ustr}}}{M_v c_{\text{ustr}} + M_a \left(\frac{p_A}{RT} - c_{\text{ustr}} \right)}$$

Finally, choose **Vapor mass fraction** in the **Specify** list to set a value or expression for the **Upstream vapor mass fraction** ω_{ustr} . The vapor concentration is then defined as

$$c_{\text{ustr}} = \frac{M_a \omega_{\text{ustr}}}{M_v (1 - \omega_{\text{ustr}}) + M_a \omega_{\text{ustr}}} \cdot \frac{p_A}{RT}$$



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

Equilibrium Moisture Transport in Porous Media > Flow Conditions > Inflow

Nonequilibrium Moisture Transport in Porous Media > Flow Conditions > Inflow

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Flow Conditions > Inflow

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Inflow

Physics tab with **Hygroscopic Porous Medium > Moist Air** selected in the model tree:

Attributes > 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:

$$-\mathbf{n} \cdot \mathbf{g}_w = 0$$

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.

Note that on the boundaries of **Building Material** features and **Hygroscopic Porous Medium** features with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, the moisture flux contains both the vapor diffusive flux and the liquid capillary flux:

$$-\mathbf{n} \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = 0$$

It can be applied on exterior boundaries only.



When this node is added as a subnode of a **Liquid Water (Hygroscopic Porous Medium)** or a **Moist Air (Hygroscopic Porous Medium)** node, it only constrains the liquid capillary flux or the vapor diffusive flux respectively.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Insulation

Moisture Transport in Building Materials > Insulation

Equilibrium Moisture Transport in Porous Media > Insulation

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water > Insulation

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Insulation

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Insulation

Physics tab with **Hygroscopic Porous Medium > Liquid Water** or **Hygroscopic Porous Medium > Moist Air** selected in the model tree:

Attributes > Insulation

Liquid Water Content

Use this node to specify the moisture conditions (liquid water saturation) on exterior boundaries.

LIQUID WATER CONTENT

The equation for this condition is $s_l = s_{l,0}$, where $s_{l,0}$ is the prescribed liquid water saturation on the boundary.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Hygroscopic Porous Medium > Liquid Water > Liquid Water Content

Moisture Transport in Building Materials > Hygroscopic Porous Medium > Liquid Water > Liquid Water Content

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water > Liquid Water Content

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium Liquid Water > Liquid Water Content

Ribbon

Physics tab with **Hygroscopic Porous Medium > Liquid Water** selected in the model tree:

Attributes > Liquid Water Content

Liquid Water Flux

Use this node to add liquid water flux across exterior boundaries. A positive liquid water flux adds moisture to the domain.

LIQUID WATER FLUX

The equation for this condition is $-\mathbf{n} \cdot \mathbf{g}_l = g_0$, where g_0 is the prescribed normal liquid water flux on the boundary. It is added as a liquid capillary flux.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Hygroscopic Porous Medium > Liquid Water > Liquid Water Flux

Moisture Transport in Building Materials > Hygroscopic Porous Medium > Liquid Water > Liquid Water Flux

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water > Liquid Water Flux

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water > Liquid Water Flux

Ribbon

Physics tab with **Hygroscopic Porous Medium > Liquid Water** selected:

Attributes > Liquid Water Flux

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, Equation 7-4, Equation 7-5, and Equation 7-6 of either a [Moist Air \(Moisture Transport Interface\)](#) node or a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, 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 either a [Moist Air \(Moisture Transport Interface\)](#) node or a [Hygroscopic Porous Medium](#) 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

$$g_{\text{evap}} = \begin{cases} M_v K (c_{\text{sat}} - c_v) & \text{if } c_v > c_{\text{sat}} \text{ or } c_l > 0 \\ 0 & \text{otherwise} \end{cases}$$

where M_v is the molar mass of water vapor (SI unit: kg/mol), K is the evaporation rate factor (SI unit: m/s), c_{sat} is the saturation concentration of vapor (SI unit: mol/m^3), c_v is the vapor concentration (SI unit: mol/m^3), and c_l is the liquid water concentration on surface (SI unit: mol/m^2).

The liquid water concentration on surface is computed by solving the following equation:

$$M_v \frac{\partial c_l}{\partial t} = -g_{\text{evap}}$$

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 evaporation flux is accounted for through the moisture flux. On boundaries of a moist air domain it gives:

$$-\mathbf{n} \cdot \mathbf{g}_w = g_{\text{evap}}$$

On boundaries of a porous medium, the moisture flux contains both the vapor diffusive flux and the liquid capillary flux:

$$-\mathbf{n} \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = g_{\text{evap}}$$

The latent heat source q_{evap} (SI unit: W/m²) 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.
	<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

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Evaporation/Condensation > Moist Surface
Moisture Transport in Building Materials > Evaporation/Condensation > Moist Surface
Equilibrium Moisture Transport in Porous Media > Evaporation/Condensation > Moist Surface
Nonequilibrium Moisture Transport in Porous Media > Evaporation/Condensation > Moist Surface

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:
Boundaries > interface > Moist Surface

Moisture Content

Use this node to specify the moisture conditions on exterior boundaries. This node always constrains the vapor concentration, but different inputs (relative humidity and temperature, vapor concentration, or vapor mass fraction) can be specified.

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, Contact, and Sector Symmetry Pairs](#) in the *COMSOL Multiphysics Reference Manual* for more details.

MOISTURE CONTENT

The equation for this condition is

$$c_v = c_0$$

The vapor concentration c_0 , the relative humidity $\phi_{w,0}$, and the vapor mass fraction ω_0 are always defined at the boundary. They have different expressions depending on the chosen inputs.

When **Specify** is set to **Relative humidity**, the vapor concentration c_0 is defined as

$$c_0 = \phi_{w,0} c_{\text{sat}}(T_0)$$

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**). The vapor mass fraction is defined as

$$\omega_0 = \frac{M_v c_0}{M_v c_0 + M_a \left(\frac{p_A}{RT} - c_0 \right)}$$

Alternatively, choose **Vapor concentration** in the **Specify** list and set directly the **Vapor concentration** c_0 . The relative humidity on the boundary is then deduced

$$\phi_{w,0} = \frac{c_0}{c_{\text{sat}}(T)}$$

and the vapor mass fraction as well

$$\omega_0 = \frac{M_v c_0}{M_v c_0 + M_a \left(\frac{p_A}{RT} - c_0 \right)}$$

Finally, choose **Vapor mass fraction** in the **Specify** list to set directly ω_0 . The vapor concentration on the boundary is then deduced

$$c_0 = \frac{M_a \omega_0}{M_v (1 - \omega_0) + M_a \omega_0} \cdot \frac{p_A}{RT}$$

and the relative humidity as well

$$\phi_{w,0} = \frac{c_0}{c_{\text{sat}}(T)}$$

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default), **Weak constraints** or **Nitsche constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation. The **Nitsche constraints** option ensures continuity of the diffusive flux.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	<i>Heat and Moisture Transport in a Semi-Infinite Wall</i> : 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**
- Equilibrium Moisture Transport in Porous Media > Moisture Content**
- Nonequilibrium Moisture Transport in Porous Media > Moisture Content**
- Moisture Transport in Air > Pairs > Moisture Content**
- Moisture Transport in Building Materials > Pairs > Moisture Content**
- Equilibrium Moisture Transport in Porous Media > Pairs > Moisture Content**
- Nonequilibrium Moisture Transport in Porous Media > Pairs > Moisture Content**

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface 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.

On boundaries of a moist air domain, the moisture flux is added as a vapor diffusive flux:

$$-\mathbf{n} \cdot \mathbf{g}_w = g_0$$

On boundaries of a [Building Material](#) node or a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, the moisture flux contains both the vapor diffusive flux and the liquid capillary flux:

$$-\mathbf{n} \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = g_0$$

MOISTURE FLUX

Select the **Flux type** from the list: **General moisture flux** (the default), **Convective moisture flux, concentrations difference**, or **Convective moisture flux, pressures difference**.

General Moisture Flux

It adds g_0 to the total flux across the selected boundaries. Enter a positive value for g_0 to represent a moisture flux that enters the domain.

Convective Moisture Flux, Concentrations Difference

Several options are available to specify the **Moisture transfer coefficient** h_m .

- The default option is to enter a **User defined** value for h_m .
- With the **From heat and moisture transport analogy** option, the heat transfer coefficient h to be multiplied by the analogy factor to obtain the moisture transfer coefficient h_m should be specified. It can be **User defined** (in this case the **Absolute pressure**, p_A , the **External temperature**, T_{ext} , and the **External relative humidity**, $\phi_{w,\text{ext}}$ 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**, $\phi_{\text{w,ext}}$. Note that the relative humidity at the boundary is provided by the physics interface.

For all options except **User defined**, select the **Specify the exponent in the analogy factor** checkbox to set the variable n in [Equation 4-200](#).

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**, $\phi_{\text{w,ext}}$, and the **External temperature**, T_{ext} (used for the computation of the vapor saturation pressure). For **User defined**, enter values or expressions. Else, select an **Ambient relative humidity** and an **Ambient temperature** defined in an **Ambient Properties** node under **Definitions**. Convective moisture flux is defined by
$$g_0 = \beta_p(\phi_{\text{w,ext}}p_{\text{sat}}(T_{\text{ext}}) - \phi_{\text{w}}p_{\text{sat}}(T)).$$
- If **Partial vapor pressure** is selected, enter an **External partial vapor pressure**, $p_{\text{v,ext}}$. Convective moisture flux is defined by
$$g_0 = \beta_p(p_{\text{v,ext}} - \phi_{\text{w}}p_{\text{sat}}(T)).$$

EVAPORATION

Select the **Contributes to evaporation flux** checkbox to define the moisture flux as an evaporation flux, and to contribute to the total latent heat source `mt.q_evaptot`.

	This checkbox is useful when a Heat and Moisture multiphysics coupling node is active, the resulting latent heat source is then added as a boundary heat source into the heat transfer equation.
	<i>Condensation Risk in a Wood-Frame Wall</i> : Application Library path <code>Heat_Transfer_Module/Buildings_and_Constructions/wood_frame_wall</code>

LOCATION IN USER INTERFACE

Context Menus

- Moisture Transport in Air > Moisture Flux**
- Moisture Transport in Building Materials > Moisture Flux**
- Equilibrium Moisture Transport in Porous Media > Moisture Flux**

Nonequilibrium Moisture Transport in Porous Media > Moisture Flux

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Moisture Flux

Nonequilibrium Boundary

This node is active by default on all the interior and exterior boundaries of the computational domain, adjacent to domains where a [Hygroscopic Porous Medium](#) node is applied with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation**. It shows which boundaries are adjacent to domains where both the relative humidity ϕ_w and the liquid water saturation s_l are solved.

On these boundaries, the conditions available at the physics interface level are not applicable, because they define conditions which either act only on the relative humidity, or also have an effect on the liquid water saturation. In the first case, this can be done adding the conditions to the [Moist Air \(Hygroscopic Porous Medium\)](#) subnode, and the second case is not allowed because conditions on liquid water saturation should be handled by the [Liquid Water \(Hygroscopic Porous Medium\)](#) subnode and its children. There is however one exception on interior boundaries: [Porous Interface](#).

Concerning the domain conditions, the [Moisture Source](#) and [Initial Values](#) nodes, available at the physics interface level, are not applicable in the domains delimited by the **Nonequilibrium Boundary** condition.

This node is automatically added and removed when necessary.

Open Boundary

Use this condition to define a boundary as the limit between a moist air (or hygroscopic porous medium) domain of the geometry and the rest of the same moist air (or hygroscopic porous medium) domain that is not represented in the geometry.

At the open boundary both inflow and outflow conditions are supported. Use the [Inflow](#) and [Outflow](#) features on boundaries where only inflow or outflow conditions are expected.

For an outgoing moist airflow across the boundary, it applies a zero diffusive flux condition:

$$-\mathbf{n} \cdot \mathbf{g}_w = 0, \quad \mathbf{n} \cdot \mathbf{u} \geq 0 \quad (7-7)$$

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. This is done through a Danckwerts condition on the vapor concentration:

$$-\mathbf{n} \cdot \mathbf{g}_w = M_v(c_v - c_{ustr})\mathbf{u} \cdot \mathbf{n}, \quad \mathbf{n} \cdot \mathbf{u} < 0$$

Note that on the exterior boundaries of a porous medium, the moisture flux contains both the vapor diffusive flux and the liquid capillary flux:

$$-\mathbf{n} \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = \begin{cases} 0, & \mathbf{n} \cdot \mathbf{u} \geq 0 \\ M_v(c_v - c_{ustr})\mathbf{u} \cdot \mathbf{n}, & \mathbf{n} \cdot \mathbf{u} < 0 \end{cases}$$

Alternatively the open boundary condition can set a constraint on the vapor concentration:

$$c_v = c_{ustr}, \quad \mathbf{n} \cdot \mathbf{u} < 0 \quad (7-8)$$

Both types of conditions are based on the knowledge of the upstream moisture content (from the vapor concentration, the vapor mass fraction, or the relative humidity and the temperature) at the outer boundary of the virtual domain, but the concentration and mass fraction ones should be preferred to account for the feedback of the model moisture sources and moisture constraints on the inlet moisture profile.

Both the upstream vapor concentration c_{ustr} , and vapor mass fraction ω_{ustr} are defined on this boundary. Their expressions depend on the chosen inputs.

This node can be applied on the exterior boundaries of a computational domain in which either a [Moist Air \(Moisture Transport Interface\)](#) node, a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, or a [Moist Air \(Hygroscopic Porous Medium\)](#) subnode when it belongs to a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation** is active. In the latter case, this node only constrains the vapor flux.

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 and the vapor mass fraction can be defined:

$$c_{\text{ustr}} = \phi_{\text{w,ustr}} c_{\text{sat}}(T_{\text{ustr}})$$
$$\omega_{\text{ustr}} = \frac{M_v c_{\text{ustr}}}{M_v c_{\text{ustr}} + M_a \left(\frac{p_A}{RT} - c_{\text{ustr}} \right)}$$

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} . The vapor mass fraction is then defined as

$$\omega_{\text{ustr}} = \frac{M_v c_{\text{ustr}}}{M_v c_{\text{ustr}} + M_a \left(\frac{p_A}{RT} - c_{\text{ustr}} \right)}$$

Finally, choose **Vapor mass fraction** in the **Specify** list to set a value or expression for the **Upstream vapor mass fraction** ω_{ustr} . The vapor concentration is then defined as

$$c_{\text{ustr}} = \frac{M_a \omega_{\text{ustr}}}{M_v (1 - \omega_{\text{ustr}}) + M_a \omega_{\text{ustr}}} \cdot \frac{p_A}{RT}$$

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 and upstream vapor concentration.
- The **Nitsche constraints** option enforces the upstream vapor concentration on the boundary.

This section is not available when this node is added to the [Moist Air \(Hygroscopic Porous Medium\)](#) subnode. The **Flux (Danckwerts)** condition always applies.

	<i>Evaporative Cooling of Water</i> : Application Library path Heat_Transfer_Module/Phase_Change/evaporative_cooling
	<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

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Flow Conditions > Open Boundary

Moisture Transport in Building Materials > Flow Conditions > Open Boundary

Equilibrium Moisture Transport in Porous Media > Flow Conditions > Open Boundary

Nonequilibrium Moisture Transport in Porous Media > Flow Conditions > Open Boundary

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Flow Conditions > Open Boundary

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Open Boundary

Physics tab with **Hygroscopic Porous Medium > Moist Air** selected in the model tree:

Attributes > 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. For boundaries of a moist air domain, it gives:

$$-\mathbf{n} \cdot \mathbf{g}_w = 0$$

For boundaries of a porous medium, the sum of vapor diffusive flux and liquid capillary flux is nullified:

$$-\mathbf{n} \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = 0$$

This node can be applied on the exterior boundaries of a computational domain in which either a [Moist Air \(Moisture Transport Interface\)](#) node, a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, or a [Moist Air \(Hygroscopic Porous Medium\)](#) subnode when it belongs to a [Hygroscopic Porous Medium](#) node with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation** is active. In the latter case, this node only constrains the vapor flux.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Flow Conditions > Outflow

Moisture Transport in Building Materials > Flow Conditions > Outflow

Equilibrium Moisture Transport in Porous Media > Flow Conditions > Outflow

Nonequilibrium Moisture Transport in Porous Media > Flow Conditions > Outflow

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Flow Conditions > Outflow

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Outflow

Physics tab with **Hygroscopic Porous Medium > Moist Air** selected in the model tree:

Attributes > Outflow

Periodic Condition

Use this node to prescribe periodic conditions on moisture. Depending on the domain condition active in the adjacent domain, the constraint is applied either on relative humidity, vapor concentration or vapor mass fraction on boundary pairs. For periodic temperature and pressure conditions, all these constraints are equivalent.

When this condition is applied on the boundaries of a [Hygroscopic Porous Medium](#) domain with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation**, it also constrains the liquid saturation.

BOUNDARY SELECTION

The software automatically identifies the boundaries as either source boundaries or destination boundaries. The selection for the destination can be changed by right-clicking the **Periodic Condition** node to add a **Destination Selection** subnode.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Nitsche constraints** (default), **Pointwise constraints** or **Weak constraints**. The **Nitsche constraints** option ensures continuity of the diffusive flux. This option may work better than the **Pointwise constraints** or **Weak constraints** when the meshes of the source and destination boundaries do not coincide. Alternatively, select the **Pointwise constraints** option. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings

ORIENTATION OF SOURCE

To display this section, click the **Show More Options** button () and select **Advanced Physics Options**.

See [Orientation of Source and Destination](#) in the *COMSOL Multiphysics Reference Manual*.

MAPPING BETWEEN SOURCE AND DESTINATION

To display this section, click the **Show More Options** button () and select **Advanced Physics Options**.

See [Mapping Between Source and Destination](#) in the *COMSOL Multiphysics Reference Manual*.

LOCATION IN USER INTERFACE

Context Menus

- Moisture Transport in Air > Periodic Condition**
- Moisture Transport in Building Materials > Periodic Condition**
- Equilibrium Moisture Transport in Porous Media > Periodic Condition**

Nonequilibrium Moisture Transport in Porous Media > Periodic Condition

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Periodic Condition

Porous Interface

This node is active by default in [The Moisture Transport in Free and Porous Media Interface](#) on all the interior boundaries of the computational domain adjacent to at least one porous medium domain. Here, porous medium domain refers to [Building Material](#) domains and [Hygroscopic Porous Medium](#) domains. Use this node to manage the fluxes continuity at interfaces between domains.



It is preferable to manage the interfaces between domains with this feature rather than using no feature.

When this node is not used, the condition that applies is the continuity of the total flux. In the case of an interface between a porous medium domain and a moist air domain, this condition allows for evaporation and condensation to take place. This corresponds to the **Allow evaporation/condensation** behavior of the **Free-Porous Interface** subfeature. In the case of an interface between two porous medium domains, this condition can be different from the behavior of the **Porous-Porous Interface** subfeature, and also allows for evaporation and condensation to take place.

Note that when this node is not used, no contribution to the evaporation flux and the corresponding latent heat flux is added.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Porous Interface

Moisture Transport in Building Materials > Porous Interface

Equilibrium Moisture Transport in Porous Media > Porous Interface

Nonequilibrium Moisture Transport in Porous Media > Porous Interface

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > *interface* > **Porous Interface**

Porous-Porous Interface

This node is active on a restriction of the parent **Porous Interface** feature where boundaries are adjacent to two porous medium features. Here, porous medium features refer to the **Building Material** and the **Hygroscopic Porous Medium** features. Use this node to manage the fluxes continuity at the interface between two porous medium domains.

This condition imposes the continuity of the relative humidity ϕ_w , and the continuity of different fluxes depending on the adjacent porous domains. In **Building Material** features and **Hygroscopic Porous Medium** features with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, the continuity of the total flux is ensured since the liquid and vapor phases are solved together. For any combination of these features, the following equations stand:

$$\phi_w|_1 = \phi_w|_2$$

$$(\mathbf{g}_w + M_v c_v \mathbf{u} + \mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_1 \cdot \mathbf{n} = (\mathbf{g}_w + M_v c_v \mathbf{u} + \mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_2 \cdot \mathbf{n}$$

Since the equality only constrains the total flux, phase change can occur at the interface. This phase change is accounted for in the evaporation flux:

$$g_{\text{evap}} = [(\mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_1 - (\mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_2] \cdot \mathbf{n}_2$$

Note that this flux is not directly used in the **Moisture Transport** interfaces, but in the **Heat and Moisture** coupling, where it is multiplied by the latent heat to be accounted for as a heat flux.

When at least one of the two adjacent domains is a **Hygroscopic Porous Medium** feature with **Equilibrium between liquid and gas phases** set to **Nonequilibrium formulation**, the liquid and vapor fluxes can be constrained separately, ensuring continuity of the total flux. The condition on relative humidity still stands:

$$\phi_w|_1 = \phi_w|_2$$

$$(\mathbf{g}_w + M_v c_v \mathbf{u})|_1 \cdot \mathbf{n} = (\mathbf{g}_w + M_v c_v \mathbf{u})|_2 \cdot \mathbf{n}$$

$$(\mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_1 \cdot \mathbf{n} = (\mathbf{g}_{lc} + \rho_l \mathbf{u}_l)|_2 \cdot \mathbf{n}$$

In this case, no phase change occur, therefore:

$$g_{\text{evap}} = 0$$

In all cases, the following contribution is added by the **Heat and Moisture** coupling to the heat flux at the interface:

$$q_{\text{evap}} = -L_v g_{\text{evap}}$$

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.



When this node is added as a subnode of a [Liquid Water \(Hygroscopic Porous Medium\)](#) or a [Moist Air \(Hygroscopic Porous Medium\)](#) node, it only constrains the liquid capillary flux or the vapor diffusive flux respectively.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Symmetry

Moisture Transport in Building Materials > Symmetry

Equilibrium Moisture Transport in Porous Media > Symmetry

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Liquid Water > Symmetry

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Symmetry

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Symmetry

Physics tab with **Hygroscopic Porous Medium > Liquid Water** or **Hygroscopic Porous Medium > Moist Air** selected in the model tree:

Attributes > Symmetry

Thin Moisture Barrier

Use this node to model a discontinuous moisture content across interior or exterior 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 $(\mathbf{n} \cdot (\beta \cdot \mathbf{n}))(\phi_{w,d} - \phi_{w,u})$ and $(\mathbf{n} \cdot (\beta \cdot \mathbf{n}))(\phi_{w,u} - \phi_{w,d})$, 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

$$\beta = \mu^{-1} \frac{\delta p_{sat}}{d_s} = \frac{\delta_p p_{sat}}{d_s} = \frac{\delta p_{sat}}{s_d}$$

where δ is the vapor permeability of still air (SI unit: s), p_{sat} is the saturation pressure of water vapor (SI unit: Pa), μ is the vapor resistance factor (dimensionless), δ_p is the

vapor permeability (SI unit: s), s_d is the vapor diffusion equivalent air layer thickness (SI unit: m), and d_g 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, select **Isotropic**, **Diagonal**, or **Symmetric** tensor type, and enter values or expressions for the tensor components of 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

Equilibrium Moisture Transport in Porous Media > Thin Moisture Barrier

Nonequilibrium Moisture Transport in Porous Media > Thin Moisture Barrier

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > interface > Thin Moisture Barrier

Vapor Content

Use this node to specify the moisture conditions on exterior boundaries. This node always constrains the vapor concentration, but different inputs (relative humidity and temperature, vapor concentration, or vapor mass fraction) can be specified.

VAPOR CONTENT

The equation for this condition is

$$c_v = c_0$$

The vapor concentration c_0 , the relative humidity $\phi_{w,0}$, and the vapor mass fraction ω_0 are always available at the boundary. They have different expressions depending on the chosen inputs.

When **Specify** is set to **Relative humidity**, the vapor concentration c_0 is defined as

$$c_0 = \phi_{w,0} c_{\text{sat}}(T_0)$$

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**). The vapor mass fraction is defined as

$$\omega_0 = \frac{M_v c_0}{M_v c_0 + M_a \left(\frac{p_A}{RT} - c_0 \right)}$$

Alternatively, choose **Vapor concentration** in the **Specify** list and set directly the **Vapor concentration** c_0 . The relative humidity on the boundary is then deduced

$$\phi_{w,0} = \frac{c_0}{c_{\text{sat}}(T)}$$

and the vapor mass fraction as well

$$\omega_0 = \frac{M_v c_0}{M_v c_0 + M_a \left(\frac{p_A}{RT} - c_0 \right)}$$

Finally, choose **Vapor mass fraction** in the **Specify** list to set directly ω_0 . The vapor concentration on the boundary is then deduced

$$c_0 = \frac{M_a \omega_0}{M_v (1 - \omega_0) + M_a \omega_0} \cdot \frac{p_A}{RT}$$

and the relative humidity as well

$$\phi_{w,0} = \frac{c_0}{c_{\text{sat}}(T)}$$

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Hygroscopic Porous Medium > Moist Air > Vapor Content
Moisture Transport in Building Materials > Hygroscopic Porous Medium > Moist Air > Vapor Content
Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Vapor Content
Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Vapor Content

Ribbon

Physics tab with **Hygroscopic Porous Medium > Moist Air** selected:

Attributes > Vapor Content

Vapor Flux

Use this node to add vapor flux across exterior boundaries. A positive vapor flux adds moisture to the domain.

The moisture flux is added as a vapor diffusive flux:

$$-\mathbf{n} \cdot \mathbf{g}_w = g_0$$

VAPOR FLUX

Select the **Flux type** from the list: **General vapor flux** (the default), **Convective vapor flux, concentrations difference**, or **Convective vapor flux, pressures difference**.

General Vapor 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 Vapor Flux, Concentrations Difference

Several options are available to specify the **Vapor 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** checkbox to set the variable n in [Equation 4-200](#).

Convective Vapor Flux, Pressures Difference

Enter a value for the **Vapor 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 $g_0 = \beta_p(p_{v,\text{ext}} - \phi_w p_{\text{sat}}(T))$.

LOCATION IN USER INTERFACE

Context Menus

Moisture Transport in Air > Hygroscopic Porous Medium > Moist Air > Vapor Flux

Moisture Transport in Building Materials > Hygroscopic Porous Medium > Moist Air > Vapor Flux

Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Vapor Flux

Nonequilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air > Vapor Flux

Ribbon

Physics tab with **Hygroscopic Porous Medium** > **Moist Air** selected:

Attributes > **Vapor Flux**

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, Equation 7-4, Equation 7-5, or Equation 7-6 of either a **Moist Air (Moisture Transport Interface)** node or a **Hygroscopic Porous Medium** node with **Equilibrium between liquid and gas phases** set to **Equilibrium formulation**, 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 either a **Moist Air (Moisture Transport Interface)** node or a **Hygroscopic Porous Medium** 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 evaporation flux is accounted for through the moisture flux. On boundaries of a moist air domain it gives:

$$-\mathbf{n} \cdot \mathbf{g}_w = g_{\text{evap}}$$

On boundaries of a porous medium, the moisture flux contains both the vapor diffusive flux and the liquid capillary flux:

$$-\mathbf{n} \cdot (\mathbf{g}_w + \mathbf{g}_{lc}) = g_{\text{evap}}$$

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.
	<i>Evaporative Cooling of Water</i> : Application Library path Heat_Transfer_Module/Phase_Change/evaporative_cooling

LOCATION IN USER INTERFACE

Context Menus

- Moisture Transport in Air > Evaporation/Condensation > Wet Surface**
- Moisture Transport in Building Materials > Evaporation/Condensation > Wet Surface**
- Equilibrium Moisture Transport in Porous Media > Evaporation/Condensation > Wet Surface**
- Nonequilibrium Moisture Transport in Porous Media > Evaporation/Condensation > Wet Surface**

Ribbon

Physics tab with *interface* as **Moisture Transport in Air**, or any version of the Moisture Transport interface selected:

Boundaries > *interface* > 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** checkbox 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, $\mathbf{g}_l$:

$$\mathbf{g}_l = \rho_l \mathbf{u}_l + \mathbf{g}_{lc} = -\rho_l \frac{\kappa_{rl} \kappa}{\mu_l} (\nabla p_A - \rho_l \mathbf{g}) + \mathbf{g}_{lc}$$

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^2) 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, coupled heat transfer and moisture transport, and moisture flow modeling.

Some of these interfaces couple an interface of the Heat Transfer Module with an interface of another module (the CFD Module or AC/DC Module).

The multiphysics interfaces are found under the Heat Transfer branch (), and their availability depends on the COMSOL products available.



For a detailed overview of the functionality available in each product, visit www.comsol.com/products/specifications/

The multiphysics interfaces add [Domain Multiphysics Couplings](#) and [Boundary Multiphysics Couplings](#). They predefine the couplings through specific settings in the multiphysics couplings and in the constituent interfaces to facilitate easy set up of models. These settings are detailed in the following sections:

- [The Nonisothermal Flow and Conjugate Heat Transfer Interfaces](#)
- [The Heat Transfer with Surface-to-Surface Radiation Interface](#)

- [The Heat Transfer with Radiation in Participating Media Interface](#)
- [The Heat Transfer with Radiation in Absorbing-Scattering Media Interface](#)
- [The Heat Transfer with Radiative Beam in Absorbing Media Interface](#)
- [The Thermoelectric Effect Interface](#)
- [The Heat and Moisture Transport Interfaces](#)
- [The Moisture Flow Interfaces](#)
- [The Heat and Moisture Flow Interfaces](#)

A brief description of other multiphysics interfaces coupling an interface of the Heat Transfer Module with other interfaces is given in the following sections:

- [The Joule Heating Interface](#)
- [The Laser Heating Interface](#)
- [The Induction Heating Interface](#)
- [The Microwave Heating Interface](#)

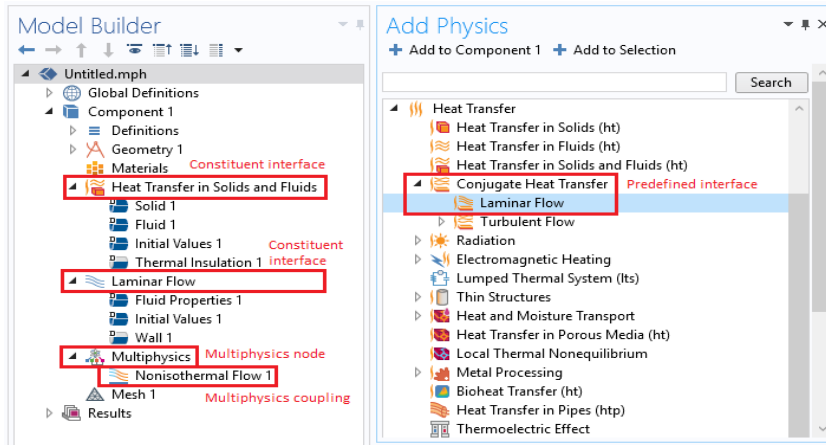
Links to thorough information about these interfaces are given in the corresponding sections.

Also:

- See [The Thermal Stress, Solid Interface](#) and [The Joule Heating and Thermal Expansion Interface](#) in the *Structural Mechanics Module User's Guide* for other multiphysics interfaces having [The Heat Transfer in Solids Interface](#) as a constituent interface. These two multiphysics interfaces (found under the Structural Mechanics branch ()) add in particular the [Thermal Expansion](#) multiphysics coupling.
- See each constituent interface documentation for more details about the common settings (in [The Heat Transfer Module Interfaces](#) for the Heat Transfer Module interfaces, and in the other modules documentation for the other interfaces).
- See [The Heat Transfer Features](#) and [The Moisture Transport Features](#) for details about the features available with the Heat Transfer Module constituent interfaces.
- See [Multiphysics Couplings](#) for details about the multiphysics couplings added with the Heat Transfer Module predefined multiphysics interfaces.

The predefined interfaces are found under the branches of the **Model Wizard** or **Add Physics** window. They add the constituent interfaces and the **Multiphysics** node containing one or more multiphysics couplings.

In the figure below, the predefined interface **Conjugate Heat Transfer, Laminar Flow** is found under the **Heat Transfer** branch. It adds the Heat Transfer in Solids and Fluids and Laminar Flow constituent interfaces, and the **Multiphysics** node containing the **Nonisothermal Flow** multiphysics coupling.



The Nonisothermal Flow and Conjugate Heat Transfer Interfaces

In this section:

- [Advantages of Using the Multiphysics Interfaces](#)
- [The Nonisothermal Flow, Laminar Flow and Turbulent Flow Interfaces](#)
- [The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces](#)
- [Settings for Physics Interfaces and Coupling Features](#)
- [Coupling Feature](#)
- [Physics Interface Features](#)
- [Preset Studies](#)

See [The Laminar Flow Interface](#), [The Turbulent Flow, Algebraic yPlus Interface](#), [The Turbulent Flow, L-VEL Interface](#), [The Turbulent Flow, k- \$\epsilon\$ Interface](#), [The Turbulent Flow, SST 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.

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** window, 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 <i>COMSOL Multiphysics Reference Manual</i> .

- 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, Realizable k-ε** interface () combines a Heat Transfer in Fluids interface with a Turbulent Flow, Realizable 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, SST** interface () combines a Heat Transfer in Fluids interface with a Turbulent Flow, SST 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, Spalart-Allmaras** interface () combines a Heat Transfer in Fluids interface with a Turbulent Flow, Spalart-Allmaras interface.
- The **Turbulent Flow, v2-f** interface () combines a Heat Transfer in Fluids interface with a Turbulent Flow, v2-f 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** window, **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 in Solids and Fluids 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, Realizable k-ε** interface () combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, Realizable 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, SST** interface () combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, SST 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, Spalart-Allmaras** interface () combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, Spalart-Allmaras interface.
- The **Turbulent Flow, v2-f** interface () combines a Heat Transfer in Solids and Fluids interface with a Turbulent Flow, v2-f 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 an 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.
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- \$\epsilon\$ Interface](#), listed in the sections [Domain](#), [Boundary](#), [Pair](#), and [Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, REALIZABLE k- ϵ

See the *CFD Module User's Guide* for a description of the available physics features for [The Turbulent Flow, Realizable k- \$\epsilon\$ 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- ω

See the *CFD Module User's Guide* for a description of the available physics features for [The Turbulent Flow, k- \$\omega\$ Interface](#), listed in the sections [Domain](#), [Boundary](#), [Pair](#), and [Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, SST

See the *CFD Module User's Guide* for a description of the available physics features for [The Turbulent Flow, SST 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- \$\epsilon\$ Interface](#), listed in the sections [Domain](#), [Boundary](#), [Pair](#), and [Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, SPALART-ALLMARAS

See the *CFD Module User's Guide* for a description of the available physics features for [The Turbulent Flow, Spalart–Allmaras Interface](#), listed in the sections [Domain](#), [Boundary](#), [Pair](#), and [Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, v2-f

See the *CFD Module User's Guide* for a description of the available physics features for [The Turbulent Flow, v2-f 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** window), 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 in Solids 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 Orbital Thermal Load Interface

In this section:

- [The Heat Transfer with Orbital Thermal Loads Multiphysics Interface](#)
- [Physics Interface Features](#)
- [Coupling Feature](#)

The Heat Transfer with Orbital Thermal Loads Multiphysics Interface

When the predefined **Heat Transfer with Orbital Thermal Loads** interface () is added (found under the **Heat Transfer** branch () of the **Model Wizard** or **Add Physics** window), it combines the Heat Transfer in Solids and the Orbital Thermal Loads interfaces to model heat transfer in a spacecraft, combined with the orbit and solar irradiation computation.

[Time Dependent](#), [Orbit Calculation](#), [Orbital Temperature](#) and [Orbit Thermal Loads](#) studies are supported in 3D.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Solids** and **Orbital Thermal Loads** 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, and thermal boundary conditions, is also available.

The Orbital Thermal Loads interface provides features to model the spacecraft orbit and positioning in order to account for thermal radiation as an energy transfer between boundaries with an external radiation source coming from the Sun.

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 Orbital Thermal Loads 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 8-3: MODIFIED SETTINGS FOR THE HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION INTERFACE.

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Orbital Thermal Loads	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	
Heat Transfer with Surface-to-Surface Radiation	The Heat Transfer in Solids and Orbital Thermal Loads 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](#).

ORBITAL THERMAL LOADS

The available physics features for [The Orbital Thermal Loads Interface](#) are listed in [Feature Nodes for the Orbital Thermal Loads 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** window), 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 8-4: 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 in Solids 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** window), 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 nonemitting.

Stationary, time-domain, and frequency-domain modeling are supported in all space dimensions.

ON THE CONSTITUENT PHYSICS INTERFACES

As a predefined multiphysics coupling, **Heat Transfer in Solids** and **Radiation in Absorbing-Scattering Media** interfaces are added to the Model Builder. In addition, a **Multiphysics** node is added, which automatically includes the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling feature.

The Heat Transfer in Solids interface provides features for modeling heat transfer by conduction, convection, and radiation in optically thick media. A **Solid** model is active by default on all domains. All functionality to include other heat transfer models, like **Fluid**, is also available.

The Radiation in Absorbing-Scattering Media interface provides features to account for absorbed and scattered radiation in a semitransparent medium. Radiative emission is not considered. The radiative intensity equations defined in participating media domains are approximated with either the Discrete Ordinates Method or the P1

Approximation. It also computes the heating due to the propagation of rays. An **Absorbing-Scattering Medium** model is active by default on all domains, and an **Opaque Surface** node is active on all boundaries. In addition, when **Discrete ordinates method** is selected, **Continuity on Interior Boundary** is also added.

AS AN ADD-ON MULTIPHYSICS COUPLING

The **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling is also available when there is at least one of each of the following interfaces with the specified model activated:

- Any version of the Heat Transfer interface, with any model
- The Radiation in Absorbing-Scattering Media interface, with an **Absorbing-Scattering Medium** model

SETTINGS FOR THE PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, specific settings are included with the physics interfaces and the coupling feature.

TABLE 8-5: 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 in Solids 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** window), 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 8-6: 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 in Solids 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** window), 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 8-7: MODIFIED SETTINGS FOR A THERMOELECTRIC EFFECT INTERFACE.

PHYSICS INTERFACE	MODIFIED SETTINGS
Electric Currents	No changes.
Heat Transfer in Solids	No changes.
Thermoelectric Effect	The Domain Selection is the same as that of the participating physics interfaces. The Boundary Selection is the same as the exterior and interior boundaries of the Domain Selection of the participating physics interfaces. But both a thermal and an electrical model need to be active on the boundaries for the coupling to be applicable. Add for example a Thin Layer node under the Heat Transfer interface and an Electric Shielding node under the Electric Currents interface to make it applicable. The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Coupled Interfaces section of the Thermoelectric Effect node's Settings window.
Electromagnetic Heating	The Domain Selection is the same as that of the participating physics interfaces. The Boundary Selection contains all the boundaries of the participating interfaces. The corresponding Electric Currents and Heat Transfer in Solids interfaces are preselected in the Coupled Interfaces section of the Thermoelectric Effect node's Settings window.

Coupling Features

See [Thermoelectric Effect](#) and [Electromagnetic Heating](#) for a description of the multiphysics couplings.



Use the online help in COMSOL Multiphysics to locate and search all the documentation. All these links also work directly in COMSOL Multiphysics when using the Help system.

Physics Interface Features

Physics nodes are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

HEAT TRANSFER IN SOLIDS

The available physics features for [The Heat Transfer in Solids Interface](#) are listed in [Feature Nodes for the Heat Transfer Interface](#).

ELECTRIC CURRENTS

The available physics features for [The Electric Currents Interface](#) are listed in [Domain, Boundary, Edge, Point, and Pair Nodes for the Electric Currents Interface](#) in the *COMSOL Multiphysics Reference Manual*.

The Heat and Moisture Transport Interfaces

In this section:

- [The Heat and Moisture Transport Multiphysics Interfaces](#)
- [The Moist Air Version of the Heat and Moisture Transport Multiphysics Interface](#)
- [The Moist Porous Media Version of the Heat and Moisture Transport Multiphysics Interface](#)
- [The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface](#)
- [Physics Interface Features](#)
- [Coupling Feature](#)

The Heat and Moisture Transport Multiphysics Interfaces

The versions of the Heat and Moisture Transport multiphysics interface are:

- [The Moist Air Version of the Heat and Moisture Transport Multiphysics Interface](#)
- [The Moist Porous Media Version of the Heat and Moisture Transport Multiphysics Interface](#)
- [The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface](#)

LATENT HEAT SOURCES

The latent heat sources are obtained by multiplying the evaporation fluxes by the latent heat of evaporation L_v , and are automatically added to the heat transfer equation:

- On boundaries, where the **Wet Surface** or **Moist Surface** features are applied, when the **Include latent heat source on surfaces** checkbox 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** window), 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 8-8: 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** window), it combines the Heat Transfer in Moist Porous Media and the Equilibrium 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 **Equilibrium 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 Equilibrium Moisture Transport in Porous Media interface provides features for modeling moisture transfer by liquid transport (capillary and convective flow), and vapor convection and diffusion. A **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 8-9: MODIFIED SETTINGS FOR THE MOIST POROUS MEDIA VERSION OF THE HEAT AND MOISTURE TRANSPORT INTERFACE.

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Moist Porous Media	In the Thermodynamics, Moist Air section of the Moist Air default subfeature of Moist Porous Medium, the Input quantity is set to Relative humidity, and the Relative humidity, ϕ_w, the Relative humidity, temperature condition, T_{ϕ_w}, and the Relative humidity, absolute pressure condition, p_{ϕ_w}, are automatically set to the variables from the Heat and Moisture multiphysics coupling feature. In the Liquid Water Properties section of the Liquid Water default subfeature of Moist Porous Medium, the Liquid water saturation, s_l, and the Velocity field, liquid water, u_l, are automatically set to the variables from the Heat and Moisture multiphysics coupling feature.
Equilibrium Moisture Transport in Porous Media	In the Model Input section of the Hygroscopic Porous Medium default feature, the Temperature, T, is automatically set to the variable from the Heat and Moisture multiphysics coupling feature.
Heat and Moisture	The Heat Transfer and Moisture Transport interfaces are preselected



Note that these settings may be overridden if another predefined coupling is added.

The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface

When the **Building Materials** version of the predefined **Heat and Moisture Transport** interface () is added (found under the **Heat Transfer** branch () of the **Model Wizard** or **Add Physics** window), 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 8-10: 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 laminar or turbulent flow () multiphysics interface is added from the **Chemical Species Transport** > **Moisture Flow** branch () of the **Model Wizard** or **Add Physics** window, one of the Single-Phase Flow interfaces (**Laminar Flow** or a 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, Realizable k-ε interface () combines a Moisture Transport in Air interface with a Turbulent Flow, Realizable 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, SST interface () combines a Moisture Transport in Air interface with a Turbulent Flow, SST 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, Spalart-Allmaras interface () combines a Moisture Transport in Air interface with a Turbulent Flow, Spalart-Allmaras interface.
- The Turbulent Flow, v2-f interface () combines a Moisture Transport in Air interface with 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 8-11: 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 μ Plus interface uses an enhanced viscosity model based on the local wall distance. The physics interface therefore includes a wall distance equation. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution.
- 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. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution.
- The Turbulent Flow, k- ϵ interface uses the standard two-equation k- ϵ model. Flow close to walls is modeled using wall functions.
- The Turbulent Flow, Realizable k- ϵ interface uses the revised 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. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.
- The Turbulent Flow, SST interface uses the Menter SST k- ω two-equation model. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.
- 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.

- The Turbulent Flow, Spalart-Allmaras interface uses the one-equation $\tilde{\nu}$ model that is designed mainly for aerodynamic applications. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.
- The Turbulent Flow, v2-f interface uses the v2-f model with realizability constraints. The v2-f model is based on the velocity fluctuations to account for turbulence anisotropy. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.

AS AN ADD-ON MULTIPHYSICS COUPLING

The **Moisture Flow** multiphysics coupling is also available when there is at least one of each of the following interfaces with the specified model activated:

- A Laminar Flow or any version of the Turbulent Flow interface, with **Fluid** model
- Any version of the Moisture Transport interface, with **Moist Air** model

Physics Interface Features for the Laminar and Turbulent Flow Versions

MOISTURE TRANSPORT IN AIR

The available physics features for [The Moisture Transport in Air Interface](#) are listed in [Feature Nodes for the Moisture Transport Interface](#) in the *Heat Transfer Module User's Guide*.

LAMINAR FLOW

The available physics features for [The Laminar Flow Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, ALGEBRAIC YPLUS

The available physics features for [The Turbulent Flow, Algebraic yPlus Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, L-VEL

The available physics features for [The Turbulent Flow, L-VEL Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, k-ε

The available physics features for [The Turbulent Flow, k-ε Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, REALIZABLE k-ε

The available physics features for [The Turbulent Flow, Realizable 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, SST

The available physics features for [The Turbulent Flow, SST 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*.

TURBULENT FLOW, SPALART-ALLMARAS

The available physics features for [The Turbulent Flow, Spalart–Allmaras 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, v2-f

The available physics features for [The Turbulent Flow, v2-f 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, Porous Media Flow () multiphysics interface is added from the **Chemical Species Transport > Moisture Flow** branch () of the **Model Wizard** or **Add Physics** window, a **Brinkman Equations** interface and a **Equilibrium 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 8-12: MODIFIED SETTINGS FOR THE MOISTURE FLOW, BRINKMAN EQUATIONS INTERFACE.

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Equilibrium 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 , u_g (Moist Air Properties section) are automatically set to the variables from the Moisture Flow multiphysics coupling feature.
Brinkman Equations	In the Porous Medium default domain feature, the Density , ρ , the Dynamic viscosity , μ (both in Fluid Properties section), the Porosity , ε , and the Permeability , κ (both in Matrix Properties section) 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.
- The relative moist air permeability κ_{rg} is set in **Equilibrium Moisture Transport in Porous Media > Hygroscopic Porous Medium > Moist Air (Moist Air Properties** section).

ON THE CONSTITUENT PHYSICS INTERFACES

The Equilibrium 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. A **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 **Porous Medium** model is active by default on all domains.

MOISTURE TRANSPORT IN POROUS MEDIA

The available physics features for [The Equilibrium 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 laminar or turbulent 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** window, a Single-Phase Flow interface (**Laminar Flow** or a turbulent flow interface), 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 8-13: MODIFIED SETTINGS FOR THE HEAT AND MOISTURE FLOW INTERFACES, LAMINAR AND TURBULENT FLOW VERSIONS.

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Moist Air	In the Model Input section of the Moist Air default domain feature, the Absolute pressure, p_A, and the Velocity field, $\mathbf{u}$, 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. In the Thermodynamics, Moist Air section of the Moist Air default feature, the Input quantity is set to Relative humidity. The Relative humidity, ϕ_w, the Relative humidity, temperature condition, T_{ϕ_w}, and the Relative humidity, absolute pressure condition, p_{ϕ_w}, are automatically set to the variables from the Heat and Moisture multiphysics coupling feature. The latent heat sources are automatically handled on boundaries where Wet Surface or Moist Surface features are applied.
Moisture Transport in Air	In the Model Input section of the Moist Air default domain feature, the Absolute pressure, p_A, and the Velocity field, $\mathbf{u}$, 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.
Laminar Flow / Turbulent Flow	In the Fluid Properties default domain feature, the Density, ρ, and the Dynamic viscosity, μ, are automatically set to the variables from the Moisture Flow multiphysics coupling feature.
Moisture Flow	The Fluid flow and Moisture Transport interfaces are preselected.
Heat and Moisture	The Heat Transfer and Moisture Transport interfaces are preselected.
Nonisothermal Flow	The Fluid flow and Heat Transfer interfaces are preselected.



Note that these settings may be overridden if another predefined coupling is added.

ON THE CONSTITUENT PHYSICS INTERFACES

The Heat Transfer in Moist Air interface provides features for modeling heat transfer by conduction, convection, and radiation. A **Moist Air** model is active by default on all domains.

The Moisture Transport in Air interface provides features for modeling moisture transport by vapor convection and diffusion. A **Moist Air** model is active by default on all domains.

The Laminar Flow interface solves the Navier–Stokes equations for conservation of momentum and the continuity equation for conservation of mass. A **Fluid Properties** model is active by default on all domains.

The different versions of the Turbulent Flow interface solve the Reynolds averaged Navier–Stokes equations for conservation of momentum, the heat transfer equation, and the continuity equation for conservation of mass. A **Fluid Properties** model is active by default on all domains. Turbulence effects are modeled in different ways:

- The Turbulent Flow, Algebraic μ Plus interface uses an enhanced viscosity model based on the local wall distance. The physics interface therefore includes a wall distance equation. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution.
- 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. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution.
- The Turbulent Flow, k- ϵ interface uses the standard two-equation k- ϵ model. Flow close to walls is modeled using wall functions.
- The Turbulent Flow, Realizable k- ϵ interface uses the revised 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. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.

- The Turbulent Flow, SST interface uses the Menter SST $k-\omega$ two-equation model. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.
- The Turbulent Flow, Low Re $k-\epsilon$ interface uses the AKN two-equation $k-\epsilon$ 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.
- The Turbulent Flow, Spalart-Allmaras interface uses the one-equation $\tilde{\nu}$ model that is designed mainly for aerodynamic applications. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.
- The Turbulent Flow, $v2-f$ interface uses the $v2-f$ model with realizability constraints. The $v2-f$ model is based on the velocity fluctuations to account for turbulence anisotropy. Flow close to walls is either modeled using wall functions or resolved down to the wall depending on local mesh resolution. The physics interface includes a wall distance equation.

Physics Interface Features for the Laminar and Turbulent Flow Versions

HEAT TRANSFER IN MOIST AIR

The available physics features for [The Heat Transfer in Moist Air Interface](#) are listed in [Feature Nodes for the Heat Transfer Interface](#). Also see [Moist Air \(Heat Transfer Interface\)](#) for details about that feature.

MOISTURE TRANSPORT IN AIR

The available physics features for [The Moisture Transport in Air Interface](#) are listed in [Settings for the Moisture Transport Interface](#). Also see [Moist Air \(Moisture Transport Interface\)](#) for details about that feature.

LAMINAR FLOW

The available physics features for [The Laminar Flow Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, ALGEBRAIC YPLUS

The available physics features for [The Turbulent Flow, Algebraic yPlus Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, L-VEL

The available physics features for [The Turbulent Flow, L-VEL Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, k-ε

The available physics features for [The Turbulent Flow, k-ε Interface](#) are listed in the section [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#) in the *CFD Module User's Guide*.

TURBULENT FLOW, REALIZABLE k-ε

The available physics features for [The Turbulent Flow, Realizable 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, SST

The available physics features for [The Turbulent Flow, SST 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*.

TURBULENT FLOW, SPALART-ALLMARAS

The available physics features for [The Turbulent Flow, Spalart–Allmaras 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, V2-F

The available physics features for [The Turbulent Flow, v2-f 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 **Porous Media 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** window, a **Brinkman Equations** interface, a **Heat Transfer in Moist Porous Media** interface, and an **Equilibrium 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 8-14: MODIFIED SETTINGS FOR THE HEAT AND MOISTURE FLOW, BRINKMAN EQUATIONS INTERFACE.

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Heat Transfer in Moist Porous Media	In the Moist Air default subfeature of Moist Porous Medium, the Absolute pressure, p_A (Model Input section) and the Velocity field, $\mathbf{u}_g$ (Heat Convection section) are automatically set to the variables from the Nonisothermal Flow multiphysics coupling feature. In the Thermodynamics, Moist Air section, the Input quantity is set to Relative humidity, and the Relative humidity, ϕ_w, the Relative humidity, temperature condition, T_{ϕ_w}, and the Relative humidity, absolute pressure condition, p_{ϕ_w}, are automatically set to the variables from the Heat and Moisture multiphysics coupling feature. In the Liquid Water Properties section of the Liquid Water default subfeature of Moist Porous Medium, the Liquid water saturation, s_l, and the Velocity field, liquid water, $\mathbf{u}_l$, are automatically set to the variables from the Heat and Moisture multiphysics coupling feature. The latent heat sources are automatically handled on boundaries where Wet Surface or Moist Surface features are applied.
Equilibrium Moisture Transport in Porous Media	In the Hygroscopic Porous Medium default domain feature, the Absolute pressure, p_A (Model Input section) and the Velocity field, $\mathbf{u}_g$ (Moist Air Properties section) are automatically set to the variables from the Moisture Flow multiphysics coupling feature. In the Model Input section, the Temperature, T, is automatically set to the variable from the Heat and Moisture multiphysics coupling feature.

TABLE 8-14: MODIFIED SETTINGS FOR THE HEAT AND MOISTURE FLOW, BRINKMAN EQUATIONS INTERFACE.

PHYSICS OR COUPLING INTERFACE	MODIFIED SETTINGS (IF ANY)
Brinkman Equations	In the Porous Medium default domain feature, the Temperature , T , and the Density , ρ are set to the variables from the Nonisothermal Flow multiphysics coupling feature. The Dynamic viscosity , μ , the Porosity , ϵ , and the Permeability , κ are set to the variable from the Moisture Flow multiphysics coupling feature. Note that the density, viscosity, and permeability are those of the moist air phase. The porosity used is the effective porosity from the point of view of moist air, it accounts for the presence of the porous matrix, and liquid water.
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 Equilibrium Moisture Transport in Porous Media interface provides features for modeling moisture transfer by liquid transport (capillary and convective flow), and vapor convection and diffusion. A **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 **Porous Medium** model is active by default on all domains.

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 Equilibrium 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:

- [Global Multiphysics Couplings](#)

- [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.

Global Multiphysics Couplings

The Heat Transfer Module has the following domain multiphysics couplings available:

- Thermal Connection, Layered Shell, Surfaces
- Thermal Connection, Nonlayered Shell



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

Thermal Connection, Layered Shell, Surfaces

Use the **Thermal Connection, Layered Shell, Surfaces** multiphysics coupling () to define a continuity condition across shared or facing surfaces, between a temperature field from a heat transfer in domain interface and a temperature field from a heat transfer in shell interface for which the **Shell type** is set to **Layered shell**.

COUPLED INTERFACES

Select the **Heat transfer** interface (any of the domain heat transfer interfaces) and **Heat transfer in shells** interface (any of the shell heat transfer interfaces) to couple.

CONNECTION SETTINGS

First select the **Connection type** among **Shared boundaries** (default) and **Facing boundaries**.

Shared boundaries corresponds to configurations where the same boundaries are shared between the **Heat transfer** and the **Heat transfer in shells** interfaces. In case the physics interfaces are defined on different boundaries, for example in case of assemblies, select the **Facing boundaries** option.

Shared Boundaries

When **Shared boundaries** is selected, the boundaries shared between the two physics interfaces are selected by default. In case the continuity condition should not be applied to the entire selection, select the **Manual control of selections** option and edit the selection in the **Boundary Selection** section above.

In the **Layered shell boundary** list select either **Automatic**, **Top**, or **Bottom** option to specify which side of the shell is connected to the solid. **Automatic** determines the side based on the boundaries normal vectors.

With the default option for **Connection tolerance**, **Automatic**, the tolerance Δ is set to 0.5% of the shell thickness, which allows for small inaccuracies on, for example, a curved geometry. By selecting **User defined**, you can modify the value of Δ .

Facing Boundaries

When **Facing boundaries** is selected, select the boundaries from the domain interface in the **Boundary Selection, Heat** section and the ones from the shell interface in the **Boundary Selection, Shell**.

In the **Layered shell boundary** list select either **Top** or **Bottom** option to specify which side of the shell is connected to the solid.

With the default option for **Connection tolerance**, **Automatic**, the tolerance Δ is set to 0.5% of the shell thickness, which allows for small inaccuracies on, for example, a curved geometry. By selecting **User defined**, you can modify the value of Δ .

Advanced

When **Connection type** is set to **Facing boundaries** the **Advanced** section is displayed. Select either **Heat transfer in shells** or **Heat transfer** (default) to define the boundaries where the continuity constraint is defined.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Thermal Connection, Layered Shell, Surfaces

when **Heat Transfer in Solids** (or another version of the Heat Transfer interface) and **Heat Transfer in Shells** (or another version of the Heat Transfer shell interface) are present.

Thermal Connection, Nonlayered Shell

Use the **Thermal Connection, Nonlayered Shell** multiphysics coupling (🔗) to define a continuity condition across an edge and a surface, shared surfaces or facing surfaces, between a temperature field from a heat transfer in domain interface and a temperature field from a heat transfer in shell interface for which the **Shell type** is set to **Nonlayered shell**.

COUPLED INTERFACES

Select the **Heat transfer** interface (any of the domain heat transfer interfaces) and **Heat transfer in shells** interface (any of the shell heat transfer interfaces) to couple.

CONNECTION SETTINGS

First select the **Connection type** among **Edges**, **Shared boundaries** (default), and **Facing boundaries**.

Select **Edges** to define continuity from an edge of a shell to a surface of a domain.

Shared boundaries corresponds to configurations where the same boundaries are shared between the **Heat transfer** and the **Heat transfer in shells** interfaces.

In case the physics interfaces are defined on different boundaries, for example in case of assemblies, select the **Facing boundaries** option.

Boundaries to Shell Edges

Using the **Edges** option, the edges for the shell interfaces and the boundaries from the domain interface are selected by default. In case the continuity condition should not be applied to the entire selection, select the **Manual control of selections** option and edit the selections in the **Boundary selection, Heat** and **Boundary selection, Shell** sections above.

In the **Shell Orientation and Position** section, set the configuration of the shell with respect to the geometry boundary by defining the offset, either to **Position — Top side on boundary**, **Midsurface on boundary**, **Bottom side on boundary**, or **User defined**.

For **User defined**, enter a value or expression for the **Relative midsurface offset**. It is given as the ratio between the offset distance and half the thickness. A value of +1 means that the actual bottom surface is located on the meshed boundary, and a value of -1 means

that the top surface is located on the meshed boundary. Values outside the range $[-1,1]$ are also allowed.

In the **Shell thickness** list select either **Automatic** (default) or **User defined** option to specify the connected area d_0 between the shell and the selected solid domain boundaries. The default is that a distance from the shell edge having the size of half the shell thickness in both perpendicular directions is connected. Select **User defined** to modify the value of d_0 .

With the default option for **Connection tolerance**, **Automatic**, the tolerance Δ is set to 0.5% of the shell thickness, which allows for small inaccuracies on for example a curved geometry. By selecting **User defined**, you can modify the value of Δ .

Shared Boundaries

When **Shared boundaries** is selected, the boundaries shared between the two physics interfaces are selected by default. In case the continuity condition should not be applied to the whole selection, select the **Manual control of selection** option and edit the selection in the **Boundary selection** section above.

In the **Layered shell boundary** list select either **Top** or **Bottom** option to specify which side of the shell is connected to the solid.

With the default option for **Connection tolerance**, **Automatic**, the tolerance Δ is set to 0.5% of the shell thickness, which allows for small inaccuracies on, for example, a curved geometry. By selecting **User defined**, you can modify the value of Δ .

Facing Boundaries

When **Facing boundaries** is selected, select the boundaries from the domain interface in the **Boundary Selection, Heat** section and the ones from the shell interface in the **Boundary Selection, Shell**.

In the **Layered shell boundary** list select either **Top** or **Bottom** option to specify which side of the shell is connected to the solid.

With the default option for **Connection tolerance**, **Automatic**, the tolerance Δ is set to 0.5% of the shell thickness, which allows for small inaccuracies on, for example, a curved geometry. By selecting **User defined**, you can modify the value of Δ .

Advanced

When **Connection type** is set to **Facing boundaries** the **Advanced** section is displayed. Select either **Heat transfer in shells** or **Heat transfer** to define the boundaries where the continuity constraint is defined.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Thermal Connection, Nonlayered Shell

when **Heat Transfer in Solids** (or another version of the Heat Transfer interface) and **Heat Transfer in Shells** (or another version of the Heat Transfer shell interface) are present.

Domain Multiphysics Couplings

The Heat Transfer Module has the following domain multiphysics couplings available:

- Electromagnetic Heating
- Heat and Moisture
- Heat Transfer with Radiation in Participating Media
- Heat Transfer with Radiation in Absorbing-Scattering Media
- Heat Transfer with Radiative Beam in Absorbing Media
- Moisture Flow
- Nonisothermal Flow
- Thermal Expansion
- Thermoelectric Effect



For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>

Electromagnetic Heating

Use the **Electromagnetic Heating** multiphysics coupling () to account for electromagnetic volumetric and surface losses in the heat equation.

See [Electromagnetic Heating](#) in the *COMSOL Multiphysics Reference Manual* for a description of this multiphysics coupling in the context of Joule heating and laser heating modeling.

See [Electromagnetic Heating](#) in the *AC/DC Module User's Guide* for a description of this multiphysics coupling in the context of induction heating modeling.

See [Electromagnetic Heating](#) in the *RF Module User's Guide* for a description of this multiphysics coupling in the context of microwave heating modeling.



- *Hepatic Tumor Ablation*: Application Library path **Heat_Transfer_Module/Medical_Technology/tumor_ablation**
- *Contact Switch*: Application Library path **Heat_Transfer_Module/Thermal_Contact_and_Friction/contact_switch**

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 a **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** checkbox.



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: $\text{W}/(\text{m}^2 \cdot \text{sr})$), 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: $\text{W}/(\text{m}^2 \cdot \text{K}^4)$).
- T is the temperature (SI unit: K)

It also adds the radiative heat flux on boundaries, depending on the selected discretization method. See [Opaque Surface \(Radiation in Participating Media and](#)

[Radiation in Absorbing-Scattering Media Interfaces](#)) for the definition of the radiative heat flux.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the name string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `htrpm1`.

COUPLED INTERFACES

This section defines the physics involved in the **Heat Transfer with Radiation in Participating Media** multiphysics coupling.

Select the **Heat transfer** interface associated to the temperature dependent variable. Select the **Radiation in participating media** interface associated to the radiative intensity variable (with Discrete Ordinates Method) or the incident radiation variable (P1 approximation).



In 1D, 1D-axi, and 2D, when the **Heat Transfer with Radiation in Participating Media** multiphysics coupling is active while solving, it disables all the out of plane features ([Out-of-Plane Heat Flux](#), [Out-of-Plane Radiation](#), [Thickness](#), [Cross Section](#)) from the heat transfer interface that is selected in the multiphysics coupling. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Heat Transfer with Radiation in Participating Media

when a Heat Transfer interface or a Slip Flow interface with any domain feature is added together with the Radiation in Participating Media interface with **Radiation in Participating Media** feature.

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 nonemitting. The radiative intensity equations are approximated either by the Discrete Ordinates Method or the P1 Approximation, depending on the **Radiation discretization method** selected in the **Participating Media Settings** section of the coupled Radiation in Absorbing-Scattering Media interface.

It adds the radiative heat source term Q (SI unit: W/m^3) to the heat transfer equation, defined by:

$$Q = \kappa G$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- G is the incident radiation (SI unit: W/m^2).

It also adds the radiative heat flux on boundaries, depending on the selected discretization method. See [Opaque Surface \(Radiation in Participating Media and Radiation in Absorbing-Scattering Media Interfaces\)](#) for the definition of the radiative heat flux.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (`_`) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `htrasm1`.

COUPLED INTERFACES

This section defines the physics involved in the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling.

Select the **Heat transfer** interface associated to the temperature dependent variable.
Select the **Radiation in absorbing-scattering media** interface associated to the radiative

intensity variable (with Discrete Ordinates Method) or the incident radiation variable (P1 approximation).



In 1D, 1D-axi, and 2D, when the **Heat Transfer with Radiation in Absorbing-Scattering Media** multiphysics coupling is active while solving, it disables all the out of plane features ([Out-of-Plane Heat Flux](#), [Out-of-Plane Radiation](#), [Thickness](#), [Cross Section](#)) from the heat transfer interface that is selected in the multiphysics coupling. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Heat Transfer with Radiation in Absorbing-Scattering Media

when a heat transfer interface with any domain feature is added together with the Radiation in Absorbing-Scattering Media interface with **Absorbing-Scattering Medium** feature.

Heat Transfer with Radiative Beam in Absorbing Media

Use the **Heat Transfer with Radiative Beam in Absorbing Media** multiphysics coupling () to model heat transfer by conduction, convection, and radiation in absorbing (semitransparent) media. The attenuation of an incident beam intensity within a semitransparent material due to absorption is determined using the Beer–Lambert Law. As the beam is absorbed it deposits energy which acts as a heat source.

It adds the radiative heat source term Q (SI unit: W/m^3) to the heat transfer equation, defined by:

$$Q = \sum_i \kappa I_i$$

where

- κ is the absorption coefficient (SI unit: m^{-1}).
- I_i is the 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.

Moisture Flow

Use the **Moisture Flow** multiphysics coupling () to simulate fluid flows in free and porous media, where the fluid properties (density, viscosity) depend on moisture content. Models can also include moisture transport in building materials. The physics interface supports low Mach numbers (typically less than 0.3).

The Moisture Flow interface solves for conservation of vapor concentration, mass, and momentum in air. It synchronizes the features from the **Moisture Transport** and **Fluid**

Flow interfaces when a turbulent flow regime is defined. It also complements the **Wall**, **Screen**, and **Interior Fan** features from the flow interface to account for moisture effects.



When considering a 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, porosity, and permeability) are the moist air properties. The porosity set through the **Moisture Flow** coupling in the **Fluid Flow** interface accounts for the presence of the liquid water phase in the porous medium. It is done by multiplying the dry material porosity by $(1 - \text{mt.s1})$, where the `mt.s1` variable is the liquid water saturation defined in the **Moisture Transport** interface. In the same way, the permeability set in the **Fluid Flow** interface is the dry material permeability multiplied by the relative moist air permeability κ_{rg} . This property can be set in the **Moist Air** subfeature of the **Hygroscopic Porous Medium** feature. In addition, the mass source or sink in the gas phase, due to evaporation or condensation, is handled by the **Moisture Flow** coupling node. This moisture source term, stored in the `mt.G_evap` variable, cancels out with the condensation term in the equilibrium formulation that couples the gas and liquid phases, but has to be accounted for in the fluid flow equations for the gas phase. Other vapor sources, that may be defined in a **Moisture Source** node and stored in the `mt.Gv` variable, are not accounted for, and should be manually added in the fluid flow interface if needed. See [Moisture Flow in Hygroscopic Porous Medium domains](#) 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 `mf1`.

DOMAIN SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific domains to define the moisture flow, or select **All domains** as needed.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Fluid flow** and **Moisture Transport** lists include all applicable physics interfaces.

The default values depend on how this coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Moisture Transport in Air** is deleted — then the **Moisture Transport** list defaults to **None** as there is nothing to couple to.

MOISTURE TRANSPORT AT WALLS

This section is available for free and porous media flows, when the **Mixture type for moist air** is set to **Concentrated species** in the moisture transport interface.

By selecting the **Account for Stefan velocity at walls** checkbox, you add a velocity contribution to the fluid flow due to the vapor flux at walls.

When prescribing a vapor flux $\mathbf{g}_w$ at boundaries in the moisture transport interface, using the **Moisture Content**, **Moisture Flux**, **Wet Surface**, or **Moist Surface** features, the **Moisture Flow** coupling node computes the Stefan velocity u_{Stefan} and applies it in the **Wall** features of the fluid flow interface, at the corresponding boundaries.

The Stefan velocity is defined as

$$\rho_g u_{\text{Stefan}} = \mathbf{n} \cdot \mathbf{g}_w$$

where ρ_g is the moist air density.

At low temperature conditions, the vapor content in moist air stays relatively small, even for high relative humidity, and the Stefan flow may be neglected. By default, the checkbox is not selected.

MOISTURE TRANSPORT TURBULENCE MODEL

This section is available for free media flow, when the fluid flow interface uses a turbulence model. In this case, the diffusive moisture flux is defined as

$$\mathbf{g}_w = -M_v(D + D_T)\nabla c_v$$

to account for the turbulent mixing caused by the eddy diffusivity in moisture convection, with the turbulent diffusivity defined as

$$D_T = \frac{\nu_T}{Sc_T}$$

where ν_T is defined by the flow interface, and the turbulent Schmidt number Sc_T depends on the **Moisture transport turbulence model**.

Select an option from the **Moisture transport turbulence model** list: **Kays–Crawford** (the default) or **User-defined turbulent Schmidt number**.

For **User-defined turbulent Schmidt number**, enter a **Turbulent Schmidt number** Sc_T (dimensionless). The turbulent kinematic viscosity ν_T is taken directly from the fluid flow interface.

For **Kays–Crawford**, see [Kays–Crawford Model for Turbulent Diffusivity](#) for details about the definition of Sc_T .

The **Turbulence model type** used by the fluid flow interface can be displayed by selecting the **Show or Hide Physics Property Settings** button at the right of the **Fluid flow** list.



When an interface is selected from the **Moisture Transport** list, some of its model inputs are forced with values from the **Moisture Flow** node. In addition, it defines how the turbulence has to be accounted for, depending on the **Fluid flow** interface's turbulence settings. Therefore, each moisture transport or fluid flow interface should be used in at most one **Moisture Flow** node. In cases where multiple fluid flow interfaces are used, an equal number of moisture transport interfaces and **Moisture Flow** nodes are needed to define proper multiphysics couplings.

	If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the Fluid flow or Moisture Transport lists. This behavior is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See Multiphysics Modeling Workflow in the <i>COMSOL Multiphysics Reference Manual</i>.
	<i>Evaporative Cooling of Water</i>: Application Library path Heat_Transfer_Module/Phase_Change/evaporative_cooling <i>Drying of a Potato Sample</i>: Application Library path Heat_Transfer_Module/Phase_Change/potato_drying

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Moisture Flow

- when any version of a **Single-Phase Flow** (or **Brinkman Equations**, or **Free and Porous Media Flow, Brinkman**) 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**, or **Free and Porous Media Flow, Brinkman**, or **Darcy's Law**) interface with **Porous Medium** 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 interface has no effect if the multiphysics coupling stabilization is active but remains active if not. However, the isotropic diffusion is not a multiphysics stabilization and is controlled by each physics interface. Finally, when one of the physics interfaces or the multiphysics coupling is not solved in a study step, then the stabilization of each solved physics is used instead of the coupled stabilization, and the solver suggestions are uncoupled.



The pressure, velocity, and temperature variables of the **Nonisothermal Flow** coupling node are set to the **Common Model Input** values of the **Default Model Inputs** node on its complementary selection, that is, all domains except those from the **Selection** list. It allows to couple multiple fluid flow interfaces with a single heat transfer interface. See [Default Model Inputs](#) in the *COMSOL Multiphysics Reference Manual* for details.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (`_`) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `nitf1`.

DOMAIN SELECTION

The **Nonisothermal Flow** coupling node selection is locked to the union of

- the intersection of the **Fluid** or **Moist Air** features' selection with the **Fluid Properties** feature's selection
- the intersection of the **Porous Medium** feature's selection with the **Fluid and Matrix Properties** feature's selection

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Fluid flow** and **Heat transfer** lists include all applicable physics interfaces.

The default values depend on how this coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Heat Transfer in Fluids** is deleted — then the **Heat transfer** list defaults to **None** as there is nothing to couple to.

HEAT TRANSFER TURBULENCE MODEL

This section is available when the fluid flow interface uses a turbulence model. Select an option from the **Heat transport turbulence model** list: **Kays–Crawford** (the default), **Extended Kays–Crawford**, or **User-defined turbulent Prandtl number**.

For **Extended Kays–Crawford**, enter a **Reynolds number at infinity** 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** checkbox 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.

ENERGY BALANCE

The **Include viscous dissipation** checkbox 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 <i>COMSOL Multiphysics Reference Manual</i>.
	<i>Heat Sink</i>: 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:

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

The term $\mathbf{J}_{e,s} = -\sigma_s S_s \nabla_t T$ is also added to the current density, which is then defined as:

$$\mathbf{J} = -\sigma_s(\nabla_{\mathbf{t}} V + S_s \nabla_{\mathbf{t}} T)$$

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the **name** string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is **tee**.

DOMAIN SELECTION

From the **Selection** list, choose the domains where thermoelectric effect should be applied. Only domains where both electrical and thermal models are active can be selected.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries where thermoelectric effect should be applied. Only boundaries where both electrical and thermal models are active can be selected.

For the thermal model, you can choose from

- any version of the Heat Transfer interface with a **Thin Layer**, a **Thin Film**, or a **Fracture** feature active on the boundaries.
- any version of the Heat Transfer in Shells interface.

For the electrical model, you can choose from

- any version of the Electric Currents interface with an **Electric Shielding** feature active on the boundaries.
- any version of the Electric Currents, Shells interface.

COUPLED INTERFACES

This section defines the physics involved in the thermoelectric effect multiphysics coupling.

Select the **Heat Transfer** interface associated to the temperature dependent variable. Select the **Electromagnetic** interface associated to the electric potential dependent variable.

MODEL INPUT

This section contains fields and values that are inputs for expressions defining material properties. If such user-defined property groups are added, the model inputs appear here.

Temperature

This section is available when material properties are temperature-dependent. By default, the temperature of the coupled heat transfer interface is used and the section is not editable. To edit the **Temperature** field, click **Make All Model Inputs Editable** (). The available options are **User defined** (default), **Common model input** (the `minput.T` variable, set to 293.15 K by default) and all temperature variables from the physics interfaces included in the model. To edit the `minput.T` variable, click the **Go to Source** button (), and in the **Default Model Inputs** node under **Global Definitions**, set a value for the **Temperature** in the **Expression for remaining selection** section.

COORDINATE SYSTEM SELECTION

Select a coordinate system from the **Coordinate system** list for the interpretation of directions in anisotropic material properties. The default is the **Global coordinate system**, and the list contains any additional coordinate system (except boundary coordinate systems) added under the **Definitions** node.

See [Coordinate Systems](#) in the *COMSOL Multiphysics Reference Manual* for more details.

THERMOELECTRIC PROPERTIES

The Seebeck coefficients S and S_s (SI unit: V/K) in the domain and on the boundaries should be set.

	<i>Thermoelectric Leg</i> : 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](#)
- [Thermal Expansion, Layered](#)
- [Marangoni Effect](#)
- [Thermal Connection, Layered Shell, Edges](#)
- [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** checkbox 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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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 <i>COMSOL Multiphysics Reference Manual</i> for details on the definition of layered materials.
	See Electromagnetic Heating in the <i>COMSOL Multiphysics Reference Manual</i> for a description of the corresponding multiphysics coupling in domains, in the context of Joule heating and laser heating modeling.

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Electromagnetic Heating, Layered Shell

when any version of the Heat Transfer interface with the **Thin Layer**, **Thin Film**, or **Fracture** node, or any version of the Heat Transfer in Shells interface with the **Solid**, **Fluid**, **Porous Medium** node, is added together with the Electric Currents, Layered Shell interface.

Heat Transfer with Surface-to-Surface Radiation

Use the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling () to account for heat transfer by radiation on boundaries, when the surrounding medium does not participate in the radiation (transparent medium).

The radiosity equation defined on boundaries where surface-to-surface radiation is enabled corresponds to the radiosity method equation.

The following radiative heat source is added to the heat transfer equation on boundaries:

$$q = \varepsilon(G - e_b(T))$$

on the side of the boundary where the radiation is defined, where ε is the surface emissivity, G is the irradiation, and $e_b(T)$ is the blackbody hemispherical total emissive power. Where the radiation is defined on both sides, the radiative heat source is defined on both sides too.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern `<name>.<variable_name>`. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the `name` string must be unique. Only letters, numbers, and underscores (`_`) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is `htrad1`.

COUPLED INTERFACES

This section defines the physics involved in the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling.

Select the **Heat transfer** interface associated to the temperature dependent variable.
Select the **Surface-to-surface radiation** interface associated to the radiosity variable.



In 1D, 1D-axi, and 2D, when the **Heat Transfer with Surface-to-Surface Radiation** multiphysics coupling is active while solving, it disables all the out of plane features ([Out-of-Plane Heat Flux](#), [Out-of-Plane Radiation](#), [Thickness](#), [Cross Section](#)) from the heat transfer interface that is selected in the multiphysics coupling. This is noticed through a warning node displayed under **Compile Equations**. The radiation interfaces assume an infinite medium in the out-of-plane direction, they are thus not compatible with these features.

HEAT TRANSFER WITH SURFACE-TO-SURFACE RADIATION

Set the default opacities of the domains surrounding the boundaries where radiation is modeled.

If **Default domain opacities** is **From heat transfer interface**, the opacity depends on the domain feature:

- The **Solid**, **Porous Medium**, **Biological Tissue**, **Building Material**, and **Shape Memory Alloy** nodes define opaque domains.
- The **Fluid** and **Isothermal Domain** nodes define transparent domains.

Note that when a Heat Transfer in Shells interface is selected under **Heat transfer** in the **Coupled Interfaces** section, no domain node is available. Therefore, the default opacity set in the Surface-to-Surface Radiation interface is used: all surrounding domains are considered as transparent.

If **Default domain opacities** is **Transparent**, all domains are considered transparent by default.



With either option the default opacity may be overridden on each domain with the [Opacity \(Surface-to-Surface Radiation and Orbital Thermal Loads Interfaces\)](#) boundary node.



The opacity setting is used when the **Emitted radiation direction** is defined by **Opacity controlled** in surface-to-surface boundary features ([Diffuse Mirror \(Surface-to-Surface Radiation Interface\)](#), [Diffuse Surface \(Surface-to-Surface Radiation Interface\)](#), [Prescribed Radiosity \(Surface-to-Surface Radiation Interface\)](#), [Opaque Surface \(Surface-to-Surface Radiation Interface\)](#), and [Specular Mirror \(Surface-to-Surface Radiation Interface\)](#)): surface-to-surface radiation propagates in nonopaque domains. Alternatively the **Emitted radiation direction** can be defined using the normal orientation or on both sides of boundaries. In this case this setting is ignored. Note that on boundaries where the [Semitransparent Surface \(Surface-to-Surface Radiation Interface\)](#) is applied, the **Emitted radiation direction** is **Both sides**, and this setting is always ignored.

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.

Thermal Expansion, Layered

Use the **Thermal Expansion, Layered** 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 [Thermal Expansion, Layered](#) in the *Structural Mechanics Module User's Guide* for more details about this multiphysics coupling.

Marangoni Effect

The **Marangoni Effect** multiphysics coupling () accounts for Marangoni convection. Marangoni convection occurs when the surface tension of a fluid-fluid interface (generally liquid-air) depends on the concentration of a species or on the temperature distribution. In the case of temperature dependence, the Marangoni effect is also called

thermo-capillary convection. It is of primary importance in the fields of welding, crystal growth, and electron beam melting of metals.

The Marangoni effect is a shear stress that depends on the tangential and normal variations of surface tension with temperature gradient. It has the following contribution described by forces induced on the fluid/fluid interface:

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

where σ is the surface tension coefficient (N/m). The first term on the right-hand-side accounts for the normal forces related to curvature effects, while the second term stands for tangential forces. Note that this formulation is intended for laminar flow regimes only.

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The **Name** is used primarily as a scope prefix for variables defined by the coupling node. Refer to such variables in expressions using the pattern <name>.<variable_name>. In order to distinguish between variables belonging to different coupling nodes or physics interfaces, the **name** string must be unique. Only letters, numbers, and underscores (_) are permitted in the **Name** field. The first character must be a letter.

The default **Name** (for the first multiphysics coupling feature in the model) is **mar1**.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Fluid flow** and **Heat transfer** lists include all applicable physics interfaces.

The default values depend on how this coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (Mac and Linux users), or context menu (all users), then the first physics interface of each type in the component is selected as the default.
- If it is added automatically when a multiphysics interface is chosen in the **Model Wizard** or **Add Physics** window, then the two participating physics interfaces are selected.

You can also select **None** from either list to uncouple the node from a physics interface. If the physics interface is removed from the **Model Builder** — for example, **Heat Transfer**

in **Fluids** is deleted — then the **Heat transfer** list defaults to **None** as there is nothing to couple to.



If a physics interface is deleted and then added to the model again, then in order to reestablish the coupling, you need to choose the physics interface again from the **Fluid flow** or **Heat transfer** lists. This is applicable to all multiphysics coupling nodes that would normally default to the once present interface. See [Multiphysics Modeling Workflow](#) in the *COMSOL Multiphysics Reference Manual*.

SURFACE TENSION

Select a **Surface tension coefficient type**: **Library coefficient**, **liquid/gas interface**, or **User defined** (the default).

- For **Library coefficient**, **liquid/gas interface** choose an option from the **Library surface tension coefficient** list.
- For **User defined** enter a **Surface tension coefficient** σ (SI unit: N/m).

CONTACT ANGLE

The attachment angle between the fluid/fluid interface and the adjacent walls can be specified in this section. It influences the curvature of the interface provided a **Deforming Domain** feature from the **Moving Mesh** interface is active on the adjacent domain.

Select an option from the **Specify contact angle** list — **Directly** (the default) or **Through Young’s equation**.

- For **Directly** enter a **Contact angle** θ_w (SI unit: rad). The default is $\pi/2$ radians.
- For **Through Young’s equation** enter values or expressions for **Phase 1-Solid surface energy density** γ_{s1} (SI unit: J/m²) and **Phase 2-Solid surface energy density** γ_{s2} (SI unit: J/m²).



Marangoni Effect: Application Library path **Heat_Transfer_Module/Tutorials,_Forced_and_Natural_Convection/marangoni_effect**

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Marangoni Effect

when any of the following interface is added together with **Heat Transfer in Fluids** (or another version of the Heat Transfer Interface):

Single-Phase Flow (any version)

Porous Media and Subsurface Flow, Brinkman Equations

Thermal Connection, Layered Shell, Edges

Use the **Thermal Connection, Layered Shell, Edge** multiphysics coupling () to define a continuity condition across an edge and a surface between a temperature field from a heat transfer in domain interface and a temperature field from a heat transfer in shell interface for which the **Shell type** is set to **Layered shell**.

EDGE SELECTION

By default the edge selection is locked and set all edges that belong to the two physics interfaces selected in the **Coupled Interfaces** section. When **Manual control of selections** option is selected in the **Connection Settings** section, it is possible to edit the edge selection.

BOUNDARY SELECTION, HEAT

By default the boundary selection is locked and set to all external boundaries that are adjacent to the edges selected in the **Edge Selection** section. When **Manual control of selections** option is selected in the **Connection Settings** section, it is possible to edit the boundary selection.

COUPLED INTERFACES

Select the **Heat transfer** interface (any of the domain heat transfer interfaces) and **Heat transfer in shells** interface (any of the shell heat transfer interfaces) to couple.

SHELL PROPERTIES

By default the continuity condition is applied all along the shell thickness. In case of a layered material with more than one layer, clear the **Use all layers** option to select a layered material and define the layers where the continuity condition is active.

CONNECTION SETTINGS

In case the continuity condition should not be applied to the entire selection, select the **Manual control of selections** option and edit the selections in the **Boundary selection, Heat** and **Boundary selection, Shell** sections above.

With the default option for **Connection tolerance**, **Automatic**, the tolerance Δ is set to 0.5% of the shell thickness, which allows for small inaccuracies on, for example, a curved geometry. By selecting **User defined**, you can modify the value of Δ .

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog.

The **Constraint** settings can be set to **Pointwise constraints** (default) or **Weak constraints**. Select **Weak constraints** to replace the standard constraints with a weak implementation.

	• Constraint Reaction Terms • Weak Constraints • Constraint Settings
	<i>Disk-Stack Heat Sink:</i> Application Library path Heat_Transfer_Module/Thermal_Contact_and_Friction/disk_stack_heat_sink

LOCATION IN USER INTERFACE

Context Menus

Multiphysics > Thermal Connection, Layered Shell, Edges

when any of the **Heat transfer** interface (any of the domain heat transfer interfaces) is added together with the **Heat transfer in shells** interface (any of the shell heat transfer interfaces).

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 $q_s = P_s \bullet \mathbf{J}$ contribution is added to the heat flux $\mathbf{q}$:

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

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** checkbox 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** checkbox. For a given **Layered Material Link** or **Layered Material Stack**, you get access to a list of checkboxes 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 <i>COMSOL Multiphysics Reference Manual</i> for details on the definition of layered materials.
	See Electromagnetic Heating in the <i>COMSOL Multiphysics Reference Manual</i> 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 56, 211, 544, 549, 572, 580, 592, 602, 615, 657, 663, 737, 842, 849, 904, 919, 1009
- absorbing medium 98
- absorbing medium (node) 519
- absorbing-scattering medium 98
- absorbing-scattering medium (node) 521
- acceleration of gravity 52, 356
- activation energy 51, 195, 565, 623
- apparent heat capacity 210
- Application Libraries window 41
- application library examples
 - ambient settings 149
 - bioheat 528
 - bioheat transfer interface 457
 - biological tissue 531, 626
 - boundary heat source 637
 - boundary thermoelectric effect 1009
 - building material 870
 - conductive thermal resistor 840
 - continuity 640
 - continuity on interior boundary 642
 - convectively enhanced conductivity 540
 - diffuse surface 653
 - electromagnetic heating 988
 - external radiation source 814
 - external terminal 844
 - fluid 547
 - heat and moisture 991
 - heat flux 664
 - heat flux (heat transfer in thin shells interface) 771
 - heat source 556
 - heat transfer in porous media 449
 - heat transfer in thin shells 460
 - highly conductive layers 234
 - inflow (moisture transport) 896
 - initial values (heat transfer in thin shells interface) 678
 - insulation/continuity (heat transfer in thin shells interface) 791, 807, 837
 - isothermal domain 570
 - isothermal domain interface 684
 - lumped system connector 688, 758
 - Marangoni effect 1017
 - moist air 573
 - moist air (moisture transport) 884, 909
 - moist surface 901
 - moisture content 903
 - moisture flow 1000
 - moisture flux 905
 - nonisothermal flow 1006
 - opaque 579
 - opaque surface 695
 - open boundary 697
 - open boundary (moisture transport) 909
 - optically thick participating medium 581
 - outflow 698
 - out-of-plane heat flux 583
 - out-of-plane radiation 585
 - periodic condition 539, 699, 704
 - phase change material 600
 - point heat source 797
 - radiation in participating media 491
 - radiation in participating media (rpm interface) 591
 - solid 620
 - symmetry 726–727
 - temperature (heat transfer in thin

- shells interface) 789
- temperature (heat transfer interface) 728
- thermal connection, layered shell, edges 1019
- thermal contact 350, 734
- thermal expansion 1007
- thermal insulation edges 792
- thin layer 745
- thin moisture barrier 916
- wet surface 921
- arterial blood temperature 58, 528
- average gas particle diameter 51
- average particle diameter 51, 198
- axisymmetric geometries 282, 479, 481, 712
- azimuth 63, 282
- azimuthal sectors 282
- B** battery layers (node) 98, 526
- beam orientation 51, 642, 673, 749
- bioheat (node) 98, 527
- bioheat transfer interface 423, 457
 - theory 193
- biological tissue (node) 98, 457, 529
- black walls 694
- blackbody radiation 671, 708
- blackbody radiation intensity 53, 292, 586, 992
- boundary conditions
 - heat equation, and 92
 - heat transfer coefficients, and 354
- boundary flux variables (heat transfer interface) 77
- boundary heat source (node) 635, 638
- boundary heat source variable 81
- boundary interface nodes 747
- boundary multiphysics nodes 923, 1011
- boundary nodes 633, 891
- boundary pair (node) 100, 638
- Boussinesq approximation 327
- Brinell hardness 53, 348
- building material (moisture transport interface) 513
- building material (node) 98
 - heat transfer interface 455–456, 531, 992
 - moisture transport interface 868, 992
- bulk velocity 356
- C** cell periodicity (node) 536
- change thickness (node) 98
- characteristic length 356
- Charron's relation 349, 733
- coefficient of thermal expansion 60, 173, 357, 610
- common settings 34
- concentration 50, 187, 549, 572
- conduction, definition 176
- conductive heat flux variable 74
- conductive heat flux vector 92
- conductive thermal resistor (node) 837
- conjugate heat transfer (multiphysics interface) 928
- conjugate heat transfer (settings) 448
- consistent stabilization (settings) 433, 441, 443
- consistent stabilization method 103
- continuity (moisture transport interface) 891
- continuity (node) 639, 641
 - heat transfer in shells 782
- continuity on interior boundary (node) 492, 501, 642
- convection, definition 176
- convection, natural and forced 355
- convective heat flux variable 78
- convective heat flux variable, cflux 75

- convective thermal resistor (node) 841
- convectively enhanced conductivity (node) 539
- Cooper-Mikic-Yovanovich (CMY) correlation 731
- Cooper-Mikic-Yovanovich (CMY) correlation 347
- cross sectional area 50, 432, 442
- cross sectional perimeter 56, 432, 442
- cross-section (node) 98, 541
- crosswind diffusion
 - definition 103
 - heat transfer, and 103
- crosswind diffusion, consistent stabilization method 103
- D**
 - Dalton's law 187
 - damage integral analysis discretization (settings) 433
 - Darcy number 51, 198
 - density, blood 62, 528
 - dependent variables (settings) 434, 441
 - deposited beam power (node) 100, 642
 - diffuse gray radiation model 477
 - diffuse mirror (node) 644
 - diffuse spectral radiation model 477
 - diffuse surface (node) 645
 - diffuse-gray surfaces 270, 273
 - diffuse-spectral surface 276
 - Dirac pulse 211
 - direct area integration, axisymmetric geometry 283
 - discrete ordinates method (DOM) 297, 492, 501, 522, 588
 - discretization (settings) 433, 441, 443, 457, 459–460, 464, 470, 507
 - dispersion tensor 51, 626
 - dispersivities, porous media 62, 626
 - documentation 40
- domain flux variables 73
- domain heat source variables 80
- domain multiphysics nodes 923, 988
- domain nodes 518, 868
- dry solid density 214, 534
- dry solid specific heat capacity 214, 534
- dry solid thermal conductivity 214, 534
- E**
 - edge nodes 769
 - effective thermal conductivity 54, 200, 205, 207, 214, 532, 576, 626
 - effective volumetric heat capacity 60, 199, 204, 214, 620
 - electric currents interface 951
 - electromagnetic heat source 57
 - electromagnetic heat source (multiphysics coupling) 978
 - electromagnetic heating (multiphysics coupling) 950, 977, 979–980, 988
 - Electromagnetic heating, layered shell (multiphysics coupling) 1011
 - elevation 282
 - emailing COMSOL 42
 - emission, radiation and 292
 - energy rates 380
 - equilibrium moisture transport in porous media interface 510
 - evaluating view factors 279
 - events timeline (node) 807
 - external radiation source (node) 809
 - External temperature 100
 - external temperature (node) 653
 - external terminal (node) 843
- F**
 - Favre average 326
 - film thickness 236
 - fluence rate calculation (node) 542
 - fluid
 - htlsh interface 735
 - node (porous medium) 548

- fluid (node) 98, 543
- Fourier's law 544, 550, 602, 619
- fracture (node) 655
- frames, conversions between 383
- frames, moving 96
- free-porous interface (moisture transport interface) 893
- frequency factor 50, 195, 565, 623
- G**
 - gap conductance 53, 348, 729
 - generate events interface (node) 815
 - geothermal heating (node) 98, 553
 - global multiphysics nodes 983
 - global nodes 802
 - Grashof number 53, 356
 - gravity 52, 356
 - gray walls 693
 - graybody radiation 275, 671, 708
 - gray-diffuse parallel plate model 349
 - ground pointing (node) 817
 - guidelines, solving surface-to-surface radiation problems 121
- H**
 - harmonic perturbation (node)
 - heat transfer interface 659
 - heat and moisture (multiphysics coupling) 958, 976, 989
 - heat and moisture flow, laminar flow (multiphysics interface) 967
 - heat and moisture flow, porous media flow (multiphysics interface) 973
 - heat and moisture flow, turbulent flow (multiphysics interface) 967
 - heat and moisture transport (multiphysics interface) 454–456, 953–954, 956
 - heat balance 177
 - heat capacity at constant pressure 50, 173
 - heat flux (node)
 - fracture subnode 772
 - heat transfer in shells interface 769
 - heat transfer interface 660
 - thin film subnode 772
 - thin layer subnode 772
 - heat flux, interface (node)
 - heat transfer in shells 750
 - heat flux, theory 377
 - heat pipe (node) 844
 - heat rate 555, 661, 663, 706, 733
 - heat rate (node) 848
 - heat rate source (node) 850
 - heat source (node) 98, 667
 - heat transfer 554, 557
 - heat transfer in shells interface 665
 - heat transfer in shells, edge condition 775
 - heat source, interface (node)
 - heat transfer in shells 747, 753
 - heat sources
 - defining as heat rate 555
 - line and point 778
 - local thermal non-equilibrium 451
 - thin layer 667
 - heat transfer coefficients
 - theory 355
 - heat transfer in building materials interface 423
 - heat transfer in films interface 424, 463
 - heat transfer in fluids (node) 447–448
 - heat transfer in fluids interface 423, 447, 941, 945, 947, 958
 - theory 183
 - heat transfer in fractures interface 424, 468
 - heat transfer in moist porous media interface 423, 454
 - theory 206
 - heat transfer in packed beds interface

- 452
- heat transfer in porous media interface
 - 423, 448–449
 - theory 199
- heat transfer in shells interface 424, 458
- heat transfer in shells interfaces 424
- heat transfer in solids (node) 448
- heat transfer in solids and fluids interface 448
- heat transfer in solids interface 423, 446, 936, 938, 941, 944, 947, 951, 958
 - theory 182
- heat transfer interface 423, 455–456
- heat transfer interfaces 423
- Heat Transfer Module 70
- heat transfer with radiation in absorbing media (multiphysics interface) 946
- heat transfer with radiation in absorbing-scattering media (multiphysics coupling) 994
- heat transfer with radiation in absorbing-scattering media (node) 994
- heat transfer with radiation in participating media (multiphysics coupling) 992
- heat transfer with radiation in participating media (node) 992
- heat transfer with radiation in participating medium (multiphysics coupling) 942, 945
- heat transfer with radiation in participating medium (multiphysics interface) 940, 943
- heat transfer with radiative beam in absorbing media (multiphysics coupling) 947, 995
- heat transfer with radiative beam in absorbing media (node) 995
- heat transfer with surface-to-surface radiation (multiphysics coupling) 936, 939, 1013
- heat transfer with surface-to-surface radiation (multiphysics interface) 934, 937
- heat transfer with surface-to-surface radiation (node) 1013
- heat transfer with surface-to-surface radiation interface 423
- heat transfer, and streamline diffusion 103
- Heaviside function 211
- hemicube, axisymmetric geometry 283
- hygroscopic porous medium (moisture transport interface) 510–511, 514
- hygroscopic porous medium (node) 992, 1000
- I**
 - immobile fluids (node) 98, 558
 - implicit event (node) 818
 - incident intensity (node) 492, 501, 670
 - inconsistent stabilization (settings) 433, 441, 443
 - inconsistent stabilization methods 104
 - induction heating (multiphysics interface) 979
 - infinite elements (node) 98
 - inflow (node) 675, 894
 - initial values
 - node (htlsh interface) 676
 - initial values (node) 446–447, 449–450, 452, 454–457, 459, 464, 469
 - heat transfer in shells interface 676
 - heat transfer interface 560–561
 - moisture transport interface 875
 - radiation in participating media interface 562
 - radiative beam in absorbing media in-

- terface 562
 - surface-to-surface radiation interface 678
- insulation (node) 896
- insulation/continuity (node) 459, 464, 469
- internal boundary heat flux variables 78
- internal energy 51, 377
- internet resources 40
- inward heat flux 92
- irreversible transformation (node) 98, 565
- isothermal domain (node) 568
- isothermal domain (settings) 436
- isothermal domain interface (node) 436, 680
- isotropic diffusion 104, 443
- J** Joule heating (multiphysics interface) 977
- K** Kays-Crawford models 330
- knowledge base, COMSOL 42
- L** laminar flow interface 928, 959, 967
- laser heating (multiphysics interface) 978
 - latent heat of evaporation 55, 215, 532
 - Latitude 135
 - latitude 282, 811
 - layer opacity (node) 684
 - layer thickness 51, 227–228, 232
 - Layered thermal expansion 1015
 - layered thermal expansion 1015
 - Legendre coefficient 50, 302, 307, 524, 590
 - line and point heat source variables 81
 - line heat source (node) 686
 - heat transfer interface 778
 - line heat source on axis (node) 686
 - heat transfer 686
 - line heat source variable 81
 - liquid water content (node) 898
 - liquid water flux (node) 898
 - liquid water source (node) 879
 - local thermal non-equilibrium (multiphysics interface) 449
 - local thermal non-equilibrium interface 450
 - theory 201
 - local thermal nonequilibrium interface 423
 - Longitude 135
 - longitude 282, 811
 - lumped system connector (node) 687
 - heat transfer in shells interface 780
 - lumped thermal nonequilibrium boundary (node) 686
 - lumped thermal system interface 473
 - lumped thermal system, interface (node)
 - heat transfer in shells 756
 - M** Marangoni effect (multiphysics coupling) 1015
 - mean effective thermal conductivity 619
 - mean effective thermal diffusivity 620
 - mechanisms of heat transfer 176
 - metabolic heat source 57, 528
 - microwave heating (multiphysics interface) 980
 - Mikic elastic correlation 348, 731
 - moist air 551, 739
 - moist air (moisture transport interface) 509, 514
 - moist air (node) 98, 571, 992
 - moist porous medium (node) 454, 574, 992
 - moist surface (node) 899
 - moisture content 60, 186, 552, 573, 739
 - moisture content (node) 901
 - moisture diffusivity 51, 314, 319, 321, 869
 - moisture flow (multiphysics coupling)

- 966, 976, 996
- moisture flow (multiphysics interface)
 - 959, 964
- moisture flow interface
 - theory 333
- moisture flux (node) 904
- moisture source (node)
 - moisture transport interface 885
- moisture storage capacity 64, 314, 868
- moisture storage function 60
- moisture transfer coefficient 60
- moisture transport in air interface 429, 509, 962, 971
- moisture transport in building materials interface 430, 513
- moisture transport in free and porous media interface 430, 514, 516
- moisture transport in porous media interface 430
- moisture transport interface 992
- moisture transport interfaces 430
- moving frames 96
- moving mesh, heat transfer, and 212, 600
- MPH-files 41
- multiphysics couplings
 - electromagnetic heating (node) 988
 - Electromagnetic heating, layered shell (node) 1011
 - heat and moisture (node) 989
 - Marangoni effect (node) 1015
 - moisture flow (node) 996
 - nonisothermal flow (node) 1000
 - thermal expansion (node) 1007
 - thermoelectric effect (node) 1007
- mutual irradiation 52, 274
- N** natural and forced convection 355
- nodes, common settings 34
- nonequilibrium boundary (node) 906
- nonequilibrium moisture transport in air interface 430
- nonequilibrium moisture transport in porous media interface 511
- nonisothermal flow (multiphysics coupling) 106, 930, 976, 1000
- nonisothermal flow (multiphysics interface) 927
- nonisothermal flow interface
 - theory 325
- normal conductive heat flux variable 77
- normal convective heat flux variable 77
- normal total energy flux variable 77
- normal total enthalpy flux variable 77
- nucleate boiling heat rate (node) 851
- Nusselt number 55, 203, 355, 540, 595, 608
- O** opaque (node) 578
- opaque surface (node) 492, 501, 688, 692
- open boundary (moisture transport interface) 906
- open boundary (node) 696
- optically thick participating medium (node) 580
- orbital parameters (node) 819
- orbital thermal loads interface 488
 - theory 284
- outflow (moisture transport interface) 909
- outflow (node) 698
- out-of-plane flux variables 76
- out-of-plane heat flux (node) 582
- out-of-plane heat transfer
 - theory 351
- thin shells theory 231
- out-of-plane inward heat flux variable 76
- out-of-plane radiation (node) 584
- out-of-plane thickness 51, 227–228, 232,

- 236, 440, 752
- P**
 - PI approximation 301, 492, 501, 522, 588
 - pair thermal contact (node) 729
 - participating media, radiative heat transfer 291
 - Pellet-fluid interface, node 594
 - Peltier effect 339, 1007, 1019
 - Pennes' approximation 193, 529
 - performance index 56, 493, 502
 - perfusion rate, blood 63, 528
 - periodic condition (moisture transport interface) 910
 - periodic condition (node) 698
 - rpm interface 700
 - phase change interface (node) 700
 - phase change interface, exterior (node) 704
 - phase change material (node) 98, 595
 - phase transitions 596
 - physical model (settings) 432, 440, 442, 447, 449, 451, 453–454
 - physics interfaces, common settings 34
 - planet properties (node) 821
 - point heat flux (node)
 - thin rod subnode 795
 - point heat source (node)
 - heat transfer 796
 - point heat source on axis (node) 797
 - point heat source variable 81
 - point nodes 795
 - points
 - heat flux 795
 - surface-to-ambient radiation 798
 - temperature 800
 - porous interface (moisture transport interface) 912
 - porous matrix
 - node (porous medium) 592, 600
 - porous medium
 - htlsh interface 655
 - porous medium (moisture transport interface) 868
 - porous medium (node) 98, 449–450, 452, 468
 - porous-porous interface (moisture transport interface) 913
 - Prandtl number 56, 203, 330, 356
 - prescribed radiosity (node) 707
 - pressure work (node) 610
- R**
 - radiation
 - axisymmetric geometries, and 282, 479, 481, 712
 - participating media 291
 - radiation branch (settings) 476
 - radiation group (node) 711
 - radiation in participating media (node)
 - rpm interface 586
 - radiation in participating media interface
 - 490
 - theory 291
 - radiation intensity, blackbody 53, 292, 586, 992
 - radiation, definition 177
 - radiative conductance 53, 349, 731, 765
 - radiative heat flux variable 78
 - radiative heat source (node) 852
 - radiative heat, theory 263
 - Radiative Intensity Source 98
 - radiative out-of-plane heat flux variable
 - 76
 - radiative source (node) 611
 - radiative thermal resistor (node) 854
 - radiative transfer equation 292
 - radiogenic heating per mass 57, 553
 - radiosity 54, 263, 265, 274, 278, 644–645, 688, 707, 719, 726, 809

- radiosity expressions 708
- radiosity method 267, 273
- ratio of specific heats 61, 545, 550–551, 598–599
- Rayleigh number 58, 356
- rayshooting, axisymmetric geometry 283
- reflectivity 62, 263–264, 295
- refractive index 55, 267, 269, 293, 295, 308, 493, 501, 580
- relative humidity 63, 186, 215, 311, 314, 318, 320, 532, 552, 573, 739, 869, 882
- Reynolds number 58, 356, 1003
 - extended Kays-Crawford 330
- Rodriguez formula 294
- S**
 - scattering, radiation and 292
 - sectors, azimuthal 282
 - Seebeck coefficient 58, 340, 1009, 1021
 - Seebeck effect 339
 - Semi-transparent surface 100
 - semi-transparent surface (node) 719
 - sensible enthalpy 175
 - Settings for the Heat Transfer in Shells Interface 437
 - shape memory alloy (node) 99, 613
 - shell thickness 227–228, 232, 236
 - solar position 282, 810
 - solid (node) 99, 446, 458, 617
 - htlsh interface 740
 - solid with translational motion (node) 99, 621
 - solver settings 113
 - solving surface-to-surface radiation
 - problems 121
 - source terms, bioheat 527
 - spacecraft axes (node) 823
 - spacecraft orientation (node) 824
 - specific heat capacity
 - biological tissue 530
 - blood 50
 - damaged tissue 50, 567, 625
 - definition 173
 - fluid 50, 543
 - fracture 659
 - phase change 598–599
 - porous media 206
 - solids 614, 618
 - thin rod 794
 - specific heat, blood 528
 - specific surface area 58, 202, 609
 - spectral band 50, 268, 542, 578, 644, 684, 688, 707, 719, 809
 - specular mirror (node) 100, 725
 - stabilization techniques
 - crosswind diffusion 103, 433, 441, 443, 507
 - isotropic diffusion 433, 441, 443, 507
 - streamline diffusion 433, 441, 443, 507
 - standard settings 34
 - stationary study 423
 - Subsurface Flow Module 554, 560
 - subsystem definition (node) 856
 - subsystem instance (node) 856
 - sun position 814
 - sun properties (node) 825
 - surface emissivity 61, 274, 584, 644, 670–671, 694, 709, 723, 733, 758, 766, 784, 798
 - surface reflectivity 62
 - Surface-to-ambient radiation 100
 - surface-to-ambient radiation (node) 723
 - fracture subnode 784
 - thin film subnode 784
 - thin layer subnode 784
 - thin rod subnode 798
 - surface-to-ambient radiation, interface (node) 758

- surface-to-surface radiation interface
 - 476
 - theory 263
 - symmetry (moisture transport interface)
 - 914
 - symmetry (node)
 - ht interface 726
 - rpm interface 726
- T**
 - technical support, COMSOL 42
 - temperature (node)
 - heat transfer in shells interface 788
 - heat transfer interface 727
 - thin rod subnode 800
 - temperature, interface (node)
 - heat transfer in shells 761, 763, 766
 - temperature, LTS interface (node) 857
 - theory
 - bioheat transfer interface 193
 - conjugate heat transfer multiphysics
 - interface 325
 - convective heat transfer correlations
 - 354
 - heat flux and balance 377
 - heat transfer coefficients 355
 - heat transfer in fluids interface 183
 - heat transfer in moist porous media interface 206
 - heat transfer in porous media interface
 - 199
 - heat transfer in solids interface 182
 - isothermal domain feature 218
 - local thermal non-equilibrium interface 201
 - material and spatial frames 381
 - moist air fluid type 186
 - moisture flow interface 333
 - nonisothermal flow 325
 - orbital thermal loads interface 284
 - out-of-plane heat transfer 351
 - radiation in participating media interface 291
 - radiative heat transfer in transparent media 263
 - surface tension coefficients 376
 - surface-to-surface radiation interface
 - 263
 - thermal contact 346
 - thermoelectric effect interface 339
 - thin rod feature 238
 - thermal capacitor (node) 857
 - thermal conductivity 54, 176
 - thermal conductivity components, thin shells 225
 - thermal conductivity supplement 50, 214, 534
 - thermal conductivity, frames and 384
 - thermal conductivity, mean effective 619
 - thermal connection (node) 803
 - thermal connection, interface-interface (node) 831
 - Thermal connection, layered shell, surfaces 983
 - Thermal connection, nonlayered shell 985
 - thermal contact (node) 729
 - theory 346
 - thermal damage (node) 622
 - thermal diffusivity 60, 357, 620
 - thermal dispersion (node) 626
 - thermal expansion (multiphysics coupling) 1007
 - thermal expansion coefficient 60, 173
 - thermal friction 349
 - thermal insulation 92
 - thermal insulation (node) 446–447, 449–450, 452, 454–457, 734, 790

- thermal insulation, edges (node) 792
- thermal mass (node) 859
- thermoelastic damping (node) 628
- thermoelectric effect (multiphysics coupling) 950, 1007
- thermoelectric effect (multiphysics interface) 948
- thermoelectric effect interface theory 339
- Thermoelectric effect, layered shell 1019
- thermoelectric module (node) 860
- thickness 432, 442
 - out-of-plane 51, 629, 752, 755, 761, 771, 777
 - shell 51
 - thin film 51
 - thin layer 51
- thickness (node)
 - heat transfer interface 629
- thin conductive layers, definition 232
- thin film (node) 735
- thin layer (node) 740
- thin layer, general 223
- thin moisture barrier (node) 915
- thin rod (node) 793
- Thomson effect 339
- time zone 282, 811
- time-dependent study 423
- total boundaries area 50
- total energy flux variable 75
- total enthalpy flux variable, thflux 75
- total heat flux variable 74
- total internal energy 51
- total normal heat flux variable 77
- total out-of-plane heat flux variable 76
- translational motion (node) 630
- transparent media refractive index 55
- turbulence modeling constant 50, 338
- turbulent conjugate heat transfer theory 326
- turbulent flow, algebraic yPlus interface 928, 959, 967
- turbulent flow, k-e interface 928, 960, 968
- turbulent flow, k-omega interface 928, 960, 968
- turbulent flow, low Re k-e interface 928, 960, 968
- turbulent flow, L-VEL interface 928, 959, 968
- turbulent flow, realizable k-e interface 928, 960
- turbulent flow, Spalart-Allmaras interface 928, 960, 968
- turbulent flow, SST interface 928, 960, 968
- turbulent flow, v2-f interface 960
- turbulent heat flux variable 74
- turbulent mixing (node)
 - moisture transport interface 888
- turbulent nonisothermal flow interfaces theory 326
- turbulent Prandtl number 56, 330, 1003
- turbulent Schmidt number 999

V

- vapor content (node) 916
- Vapor diffusion coefficient in air 51
- vapor diffusion coefficient in air 311, 318, 320, 882
- vapor flux (node) 918
- vapor mass fraction 60, 551, 573, 739
- vapor permeability 61, 215, 314, 532, 868
- vapor permeability of still air 61
- vapor resistance factor 62
- vapor saturation concentration 51, 311, 318, 320, 882
- vapor saturation pressure 215, 314, 532, 869

vapor source (node) 889

Vickers correlation coefficient 347, 732

Vickers size index 347, 732

view factors 52, 279, 645, 809

viscous dissipation (node) 631

volume reference temperature 101

volumetric heat capacity 60, 620

von Kármán constant 331

W water content 214, 535

websites, COMSOL 42

wet surface (node) 920

Y Young's modulus 51

Z zenith 61, 282