



Porous Media Flow Module

User's Guide

Porous Media Flow Module User's Guide

© 1998–2021 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 a 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/comsol-license-agreement) and may be used or copied only under the terms of the license agreement.

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

Version: COMSOL 6.0

Contact Information

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

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

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

Part number: CM024801

C o n t e n t s

Chapter 1: Introduction

About the Porous Media Flow Module	14
Why Porous Media Flow Is Important for Modeling	14
How the Porous Media Flow Module Helps Improve Your Modeling	14
The Porous Media Flow Module Physics Interface Guide	15
Common Physics Interface and Feature Settings and Nodes	20
The Porous Media Flow Module Study Availability.	20
The Liquids and Gases Materials Database	21
Where Do I Access the Documentation and Application Libraries?	21
Overview of the User's Guide	24

Chapter 2: Porous Media Flow Modeling

What Is a Porous Medium?	28
About Darcian and Non-Darcian Flow	29
Permeability Models	30
Mass and Momentum Conservation in Porous Media	34
Mass Conservation	34
Momentum Conservation	35

Chapter 3: Single- and Multiphase Flow Interfaces

The Laminar Flow and Creeping Flow Interfaces	38
The Creeping Flow Interface.	38
The Laminar Flow Interface	39
Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow	45
Fluid Properties.	45
Volume Force	47
Initial Values	47
Wall	48

Inlet	50
Outlet	52
Symmetry	55
Open Boundary	56
Boundary Stress	56
Periodic Flow Condition	58
Flow Continuity	59
Pressure Point Constraint.	59
Point Mass Source	60
Line Mass Source	60
Gravity	61
Theory for the Single-Phase Flow Interfaces	62
General Single-Phase Flow Theory	63
Compressible Flow	65
Weakly Compressible Flow	65
The Mach Number Limit	65
Incompressible Flow	66
The Reynolds Number	67
Gravity	68
Theory for the Wall Boundary Condition	69
Prescribing Inlet and Outlet Conditions	73
Fully Developed Flow (Inlet)	75
Fully Developed Flow (Outlet)	76
No Viscous Stress.	77
Normal Stress Boundary Condition	78
Pressure Boundary Condition	78
Mass Sources for Fluid Flow	81
Numerical Stability — Stabilization Techniques for Fluid Flow	83
Solvers for Laminar Flow	85
Pseudo Time Stepping for Laminar Flow Models	87
Discontinuous Galerkin Formulation.	89
Particle Tracing in Fluid Flow	89
References for the Single-Phase Flow, Laminar Flow Interfaces	90

Chapter 4: Porous Media and

Subsurface Flow Interfaces

Theory for the Darcy's Law Interface	95
About Darcy's Law	95
Darcy's Law — Equation Formulation	96
Storage Model	97
Average Linear Velocity	98
References for the Darcy's Law Interface	98
 The Darcy's Law Interface	 100
Domain, Boundary, Edge, Point, and Pair Nodes for the Darcy's Law	
Interface	102
Porous Medium.	104
Fluid	106
Porous Matrix	107
Mass Source	109
Initial Values	110
Unsaturated Porous Medium.	110
Gravity	111
Cross Section	112
Thickness	112
Porous Electrode Coupling	113
Electrode Surface Coupling	113
Pressure	114
Mass Flux	114
Line Mass Source	115
Point Mass Source.	115
Inlet	116
Symmetry	117
No Flow	117
Flux Discontinuity.	117
Outlet	118
Precipitation	119
Interior Wall.	119
Thin Barrier	119
Pressure Head	120
Hydraulic Head.	120
Atmosphere/Gauge	120
Pervious Layer	121
Well	122
Fracture Flow	123

Theory for the Richards' Equation Interface	124
About Richards' Equation	124
Retention and Permeability Relationships	126
References for the Richards' Equation Interface	129
 The Richards' Equation Interface	 131
Domain, Boundary, Edge, Point, and Pair Nodes for the Richards' Equation Interface	132
Flownet Plot.	133
 Theory for the Fracture Flow Interface	 134
About Fracture Flow	134
Equations and Inputs.	135
 The Fracture Flow Interface	 137
Domain, Boundary, Edge, Point, and Pair Nodes for the Fracture Flow Interface	138
Fluid and Fracture Properties	140
Storage Model	141
Aperture	141
Initial Values	141
 Theory for the Layered Darcy's Law Interface	 142
Gradient Handling.	142
Area Scale Factor	143
 The Layered Darcy's Law Interface	 145
Settings for the Layered Darcy's Law Interface	146
Boundary, Edge, and Point Nodes for the Layered Darcy's Law Interface .	148
Fluid and Matrix Properties	150
Initial Values	151
Richards' Equation Model	152
Storage Model	152
Poroeleastic Storage	153
Gravity	154
Mass Source	154
Boundary Interface Features	155
Pressure, Interface	155
Mass Flux, Interface	156
Inlet, Interface	157
Outlet, Interface	158
Symmetry, Interface	159

No Flow, Interface	160
Flux Discontinuity, Interface	160
Interior Wall, Interface	161
Thin Barrier, Interface	162
Edge Features	163
Line Mass Source, Interface	163
Continuity.	164
Point Features	165
Line Mass Source	165
Point Mass Source.	165
Theory for the Brinkman Equations Interface	167
About the Brinkman Equations	167
Brinkman Equations Theory	168
References for the Brinkman Equations Interface	169
The Brinkman Equations Interface	170
Domain, Boundary, Point, and Pair Nodes for the Brinkman Equations	
Interface	173
Porous Medium.	174
Fluid	174
Porous Matrix	175
Mass Source	176
Volume Force	176
Initial Values	177
Fluid Properties.	177
Gravity	178
Theory for the Free and Porous Media Flow Interface	180
Reference for the Free and Porous Media Flow Interface	180
The Free and Porous Media Flow Interface	181
Domain, Boundary, Point, and Pair Nodes for the Free and Porous	
Media Flow Interface	183
Fluid Properties.	184
Porous Medium.	185
Fluid	185
Porous Matrix	186
Mass Source	187
Volume Force	187
Initial Values	187
Gravity	187

Wall	188
----------------	-----

Chapter 5: Chemical Species Transport Interfaces

The Transport of Diluted Species Interface	192
The Transport of Diluted Species in Porous Media Interface.	196
Domain, Boundary, and Pair Nodes for the Transport of Diluted Species Interface	198
Transport Properties	200
Initial Values	202
Mass-Based Concentrations	202
Reactions	203
No Flux	204
Inflow	205
Outflow	206
Concentration	206
Flux	206
Symmetry	208
Flux Discontinuity.	208
Partition Condition	208
Periodic Condition	209
Line Mass Source	210
Point Mass Source	211
Open Boundary	211
Thin Diffusion Barrier	212
Thin Impermeable Barrier.	212
Equilibrium Reaction.	212
Surface Reactions	213
Surface Equilibrium Reaction.	214
Fast Irreversible Surface Reaction.	214
Porous Electrode Coupling	215
Reaction Coefficients	215
Electrode Surface Coupling	216
Porous Medium	216
Fluid	217
Porous Matrix	218
Dispersion	219
Unsaturated Porous Medium	220
Liquid	220
Gas	222

Adsorption	223
Volatilization	225
Species Source	226
Hygroscopic Swelling	226
Out-of-Plane Flux	227
Fracture	229
The Transport of Diluted Species in Fractures Interface	231
Boundary, Edge, Point, and Pair Nodes for the Transport of Diluted Species in Fractures Interface	233
Adsorption	234
Concentration	235
Flux	236
Fracture	236
Inflow	237
No Flux.	238
Outflow	238
Reactions	238
Species Source	239
The Moisture Transport Interfaces	240
Theory for the Transport of Diluted Species Interface	241
Mass Balance Equation	242
Equilibrium Reaction Theory.	243
Convective Term Formulation	245
Solving a Diffusion Equation Only	246
Mass Sources for Species Transport	246
Adding Transport Through Migration	248
Supporting Electrolytes	249
Crosswind Diffusion	250
Danckwerts Inflow Boundary Condition	251
Mass Balance Equation for Transport of Diluted Species in Porous Media	252
Convection in Porous Media.	253
Diffusion in Porous Media.	255
Dispersion	256
Adsorption	258
Reactions	260
Mass Transport in Fractures	261
References	262

Chapter 6: Heat Transfer Interfaces

Selecting the Right Heat Transfer Interface	267
The Mechanisms for Heat Transfer	271
Heat Transfer in Porous Media	271
Phase Change	271

Chapter 7: Multiphysics Interfaces and Couplings

Theory for the Poroelasticity Interfaces	275
Small Strain Poroelasticity	275
Darcy's Law	278
Large Strain Poroelasticity	279
Reference for the Poroelasticity Interface	281
The Poroelasticity Interfaces	282
The Poroelasticity, Solid Interface	282
Poroelasticity (Multiphysics Coupling)	283
The Poroelasticity, Large Deformation Interface	285
Theory for the Poroelasticity, Layered Shell Interface	286
The Poroelasticity, Layered Shell Interface	287
Layered Poroelasticity (Multiphysics Coupling)	289
Theory for the Multiphase Flow in Porous Media Interface	291
Pressure-Saturation Formulation	291
Mass Source	293
Boundary Conditions	293
References	294
The Multiphase Flow in Porous Media Interface	295
Multiphase Flow in Porous Media (Multiphysics Coupling).	296
Well (Multiphysics Coupling)	297

The Moisture Flow Interface	300
The Heat and Moisture Transport Interfaces	301
The Heat and Moisture Flow Interface	302
The Reacting Flow in Porous Media Interface	303
The Nonisothermal Flow, Brinkman Equations Interface	304
The Two-Phase Flow, Level Set, Brinkman Equations Interface	305

Introduction

The Porous Media Flow Module is an optional add-on package for COMSOL Multiphysics® designed to assist you to model transport processes in porous media.

This chapter introduces you to the capabilities of this module and includes a summary of the physics interfaces as well as information about where you can find additional documentation and model examples. The last section is a brief overview with links to each chapter in this guide.

- [About the Porous Media Flow Module](#)
- [Overview of the User's Guide](#)

About the Porous Media Flow Module

In this section:

- [Why Porous Media Flow Is Important for Modeling](#)
- [How the Porous Media Flow Module Helps Improve Your Modeling](#)
- [The Porous Media Flow Module Physics Interface Guide](#)
- [Common Physics Interface and Feature Settings and Nodes](#)
- [The Porous Media Flow Module Study Availability](#)
- [The Liquids and Gases Materials Database](#)
- [Where Do I Access the Documentation and Application Libraries?](#)

Why Porous Media Flow Is Important for Modeling

Porous media are encountered in many natural as well as man-made systems. The need for advanced porous media modeling spans many industries and application areas like processes in fuel cells, paper-pulp drying, food production and safety, filtration, concrete, ceramics, moisture absorbents, textiles, paint drying, polymer composites, and detergent tablets. The most well-known natural porous media involving multiphase flow and transport are soils, aquifers, and oil and gas reservoirs. However, such processes also occur in biological tissues and plants. Recently, there has been growing interest in the biomechanics of porous tissues, engineered tissues, and in-tissue drug delivery.

Many of these porous systems are extremely complex. Therefore, the traditional concepts, models, and algorithms developed for porous media in the geosciences are not directly applicable and there is a very strong need for tailored models. Multiphysics couplings are essential to capture the effects of porous media on transport processes.

How the Porous Media Flow Module Helps Improve Your Modeling

The Porous Media Flow Module is an optional package that extends the COMSOL Multiphysics modeling environment with customized physics interfaces and functionality optimized for the analysis of transport processes in porous media. It combines physics interfaces for fundamental processes and links to COMSOL Multiphysics and the other modules for structural mechanics and electromagnetics analyses. Important in industrial porous media flow, the heat transfer and chemical

transport interfaces explicitly account for physics in the liquid, solid, and gas phases and include the possibility to model thermal nonequilibrium. The physics interfaces for heat transfer also include options to automate the calculation of effective thermal properties for multicomponent systems.


The Fluid Flow branch represent a wide range of possibilities. The Richards' Equation interface describes nonlinear flow in variably saturated porous media. The options for saturated porous media include the Darcy's Law interface for slow flow and the Brinkman Equations interface where shear is nonnegligible. The Laminar Flow interface uses the Navier–Stokes equations to cover free flows, and the Fracture Flow interface is used for modeling flow along thin layers and fractures.














This module also treats the transport of chemicals and their reactions with the Chemical Species Transport branch. The Transport of Diluted Species in Porous Media interface accounts for chemical transfer in solid, liquid, and gas phases for free and porous media flows. The Porous Media Flow Module Applications Library includes a number of examples linking these physics interfaces together.

















Predefined multiphysics couplings make it possible to conveniently model poroelastic effects and multiphase transport processes.
















The Porous Media Flow Module Physics Interface Guide













The Porous Media Flow Module extends the functionality of the physics interfaces of the base package of COMSOL Multiphysics with respect to the different physical aspects that are important for modeling porous media, like heat transfer in porous media and layered materials, diffusion and reactions of chemical species in porous media, and fluid flow in porous media. It also contains predefined multiphysics couplings, for example, with structural mechanics to simulate poroelasticity or with free flow interfaces. The table below lists all the interfaces specifically available with this module.

	<p>In the <i>COMSOL Multiphysics Reference Manual</i>:</p> <ul style="list-style-type: none"> • 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 STUDY TYPE
 Chemical Species Transport				
Transport of Diluted Species ¹		tds	all dimensions	stationary; time dependent
Transport of Diluted Species in Porous Media		tds	all dimensions	stationary; time dependent
Transport of Diluted Species in Fractures		dsf	3D, 2D, 2D axisymmetric	stationary; time dependent
 Moisture Transport				
Moisture Transport in Air		mt	all dimensions	stationary; time dependent
Moisture Transport in Porous Media		mt	all dimensions	stationary; time dependent
Moisture Transport in Building Materials		mt	all dimensions	stationary; time dependent
 Moisture Flow				
Laminar Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way Moisture Flow; time dependent, one-way Moisture Flow
Porous Media Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way Moisture Flow; time dependent, one-way Moisture Flow
 Reacting Flow				
Laminar Flow, Diluted Species ¹		—	3D, 2D, 2D axisymmetric	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
 Reacting Flow in Porous Media				
Transport of Diluted Species		rfds	3D, 2D, 2D axisymmetric	stationary; time dependent
 Fluid Flow				
 Single-Phase Flow				
Creeping Flow		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
Laminar Flow		spf	3D, 2D, 2D axisymmetric	stationary; time dependent
 Multiphase Flow				
 <i>Two-Phase Flow, Level Set</i>				
Two-Phase Flow, Level Set, Brinkman Equations		—	3D, 2D, 2D axisymmetric	time dependent with initialization
 <i>Phase Transport</i>				
Phase Transport		phtr	3D, 2D, 2D axisymmetric	stationary; time dependent
Phase Transport in Porous Media		phtr	3D, 2D, 2D axisymmetric	stationary; time dependent
 Porous Media and Subsurface Flow				
Brinkman Equations		br	3D, 2D, 2D axisymmetric	stationary; time dependent
Darcy's Law		dl	all dimensions	stationary; time dependent
Layered Darcy's Law		ldl	3D	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Fracture Flow		esff	3D, 2D, 2D axisymmetric	stationary; time dependent
Richards' Equation		dl	all dimensions	stationary; time dependent
Multiphase Flow in Porous Media		—	3D, 2D, 2D axisymmetric	stationary; time dependent
Free and Porous Media Flow		fp	3D, 2D, 2D axisymmetric	stationary; time dependent
 Nonisothermal Flow				
Brinkman Equations		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way NITF; time dependent, one-way NITF
 Fluid-Structure Interaction				
 Heat Transfer				
Heat Transfer in Solids		ht	all dimensions	stationary; time dependent
Heat Transfer in Fluids ¹		ht	all dimensions	stationary; time dependent
Heat Transfer in Solids and Fluids ¹		ht	all dimensions	stationary; time dependent
 Thin Structures				
Heat Transfer in Fractures		htlsh	3D, 2D, 2D axisymmetric	stationary; time dependent; thermal perturbation, frequency domain
 Heat and Moisture Transport				
Moist Air		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Moist Porous Media		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Building Materials		—	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
 Heat and Moisture Flow				
Laminar Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way Moisture Flow; time dependent, one-way Moisture Flow
Porous Media Flow		—	3D, 2D, 2D axisymmetric	stationary; time dependent; stationary, one-way Moisture Flow; time dependent, one-way Moisture Flow
Heat Transfer in Porous Media		ht	all dimensions	stationary; time dependent
Local Thermal Nonequilibrium		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
Heat Transfer in Packed Beds		ht	all dimensions	stationary; time dependent; thermal perturbation, frequency domain
 Structural Mechanics				
Poroelectricity, Solid		poro	3D, 2D, 2D axisymmetric	stationary; time dependent
Poroelectricity, Large Deformation, Solid		poro	3D, 2D, 2D axisymmetric	stationary; time dependent
Poroelectricity, Layered Shell ²		—	3D	stationary; time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
¹ This physics interface is included with the core COMSOL package but has added functionality for this module.				
² Requires the addition of the Composite Materials Module and the Structural Mechanics Module.				

Common Physics Interface and Feature Settings and Nodes

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

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



In the *COMSOL Multiphysics Reference Manual* see [Table 2-4](#) for links to common sections and [Table 2-5](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The Porous Media Flow Module Study Availability

All the physics interfaces included with the module have the Stationary and Time Dependent preset studies available. The Heat Transfer in Fractures Interface additionally includes the Thermal Perturbation, Eigenfrequency study to compute eigenmodes and eigenfrequencies of linearized models around an equilibrium steady-state temperature field. Custom studies are also available based on the physics interface.



[Studies and Solvers](#) and [The Physics Interfaces](#) in the *COMSOL Multiphysics Reference Manual*

The Liquids and Gases Materials Database

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



For detailed information, see [Liquids and Gases Materials Library](#) and [Materials](#) in the *COMSOL Multiphysics Reference Manual*.

Where Do I Access the Documentation and Application Libraries?

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




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

THE DOCUMENTATION AND ONLINE HELP

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

Opening Topic-Based Help

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

Opening the Documentation Window

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

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/support/updates
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/community
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/video
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

The *Porous Media Flow Module User's Guide* gets you started with modeling using COMSOL Multiphysics. The information in this guide is specific to this module. Instructions how to use COMSOL Multiphysics 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 software **Help** menu.

TABLE OF CONTENTS AND INDEX

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

THE SINGLE-PHASE AND THE MULTIPHASE FLOW BRANCH

The [Single- and Multiphase Flow Interfaces](#) section describes the Laminar and Creeping Flow interfaces. The [Phase Transport Interfaces](#) are used to model the transport of multiple immiscible phases in free and porous media flow. They are described in the *CFD Module User's Guide* and just referenced here.

THE POROUS MEDIA FLOW BRANCH

The [Porous Media and Subsurface Flow Interfaces](#) chapter describes the following physics interfaces and includes the underlying theory for each physics interface at the end of the chapter.

- The Porous Media and Subsurface Flow group of interfaces estimates the pressure and velocity of fluids flowing in free systems and within the interstices of a porous medium. [Theory for the Darcy's Law Interface](#) helps you choose the right physics interface to use.
- [The Darcy's Law Interface](#) is intended for flow in porous media that is relatively slow.
- [The Richards' Equation Interface](#) is intended for variably saturated systems.
- [The Fracture Flow Interface](#) is a special application of Darcy's law for modeling flow in fractures.
- [The Layered Darcy's Law Interface](#) is intended for flow in porous media that is relatively slow.

- [The Brinkman Equations Interface](#) is intended for fast flow in porous media flow.
- [The Free and Porous Media Flow Interface](#) is intended for modeling fluid flow in systems with both free and porous media flow.

THE CHEMICAL SPECIES TRANSPORT BRANCH

The [Chemical Species Transport Interfaces](#) chapter describes the physics interfaces found under the Chemical Species Transport branch when adding a physics interface. The Transport of Diluted Species interface is used to compute the concentration field of a dilute solute in a solvent. Transport and reactions of the species dissolved in a gas, liquid, or solid can be computed.

[The Transport of Diluted Species in Porous Media Interface](#) characterizes the fate and transport of individual or multiple and interacting chemical species for systems containing fluids, solids, and gases. Theory for the physics interfaces is included at the end of the chapter.

[The Transport of Diluted Species in Fractures Interface](#) is used to model the transport of a solute species along thin fractures. The interface takes into account diffusion, dispersion, convection, and chemical reactions in fractures. The fractures are defined by boundaries in 2D and 3D, and the solute species is assumed to be diluted in a solvent. The mass transport equation solved along the fractures is the tangential differential form of the convection-diffusion-reaction equation.

[The Moisture Transport Interfaces](#) are two different interfaces that can be used to model moisture transport in building materials and moisture transport in air, respectively, taking into account the different transport mechanisms.

THE HEAT TRANSFER BRANCH

The [Heat Transfer Interfaces](#) chapter describe the group of interfaces that estimate the temperature distribution in solids, fluids, and fluid-solid systems. [The Mechanisms for Heat Transfer](#) helps you choose the physics interface to use. It includes physics interfaces to estimate effective properties in multicomponent systems. All heat transfer interfaces come with interfaces to account for a geotherm brought about through radiogenic decay.

The Heat Transfer Interface models heat transfer by conduction and convection. Surface-to-ambient radiation effects around edges and boundaries can also be included. The physics interfaces are available in 1D, 2D, and 3D and for axisymmetric models with cylindrical coordinates in 1D and 2D.

[The Heat Transfer in Porous Media Interface](#) lets you describe heat transferred both with and without flowing fluids. You can define the velocity in the convective term with any of the flow equations just mentioned or set it with an arbitrary expression. With convective heat transfer, the effective thermal properties also include an option to estimate the dispersion or spreading of heat from small-scale velocity variations

[The Local Thermal Nonequilibrium Interface](#) solves two heat transfer equations in the solid and fluid phases, to model heat transfer in porous media for which the solid and fluid temperatures are not in equilibrium..



When your license includes the Porous Media Flow Module plus the Chemical Reaction Engineering Module or Heat Transfer Module, the Heat Transfer interface also has extended features available.

THE MULTIPHYSICS COUPLING NODES

In the [Multiphysics Interfaces and Couplings](#) chapter the predefined multiphysics interfaces are introduced.

[The Poroelasticity Interfaces](#) section describes the physics interface for Biot's poroelasticity, and combines Darcy's law with solid mechanics to provide suitable settings to describe the interaction between porous media and fluids.

[The Multiphase Flow in Porous Media Interface](#) combines the functionality of the Darcy's Law and Phase Transport in Porous Media interfaces.

[The Moisture Flow Interface](#) couples moisture transport in air by vapor diffusion and convection with a laminar flow interface.

[The Heat and Moisture Transport Interfaces](#) couple heat transfer and moisture transport in building materials or in moist air, respectively.

[The Heat and Moisture Flow Interface](#) combines the functionalities of three different interfaces: Laminar flow, Moisture Transport in Air, and Heat Transfer in Moist Air.

[The Reacting Flow in Porous Media Interface](#) combines the Brinkman equations with the Transport of Diluted Species interface.

Porous Media Flow Modeling

This chapter introduces you to the modeling of flows in porous media.

In this chapter:

- [What Is a Porous Medium?](#)
- [About Darcian and Non-Darcian Flow](#)
- [Permeability Models](#)
- [Mass and Momentum Conservation in Porous Media](#)

What Is a Porous Medium?

A porous material consists of a solid structure (porous matrix) with voids (pores) filled with a liquid or gas. Porous materials are found in all sizes and across a wide range of applications - from nanomaterials to porous reactors, for cooling of electronic components to large scale geotechnical applications. What they all have in common is that the total size of the material is much bigger than the mean pore diameter, so that a macroscopic approach must be used to model the applications.

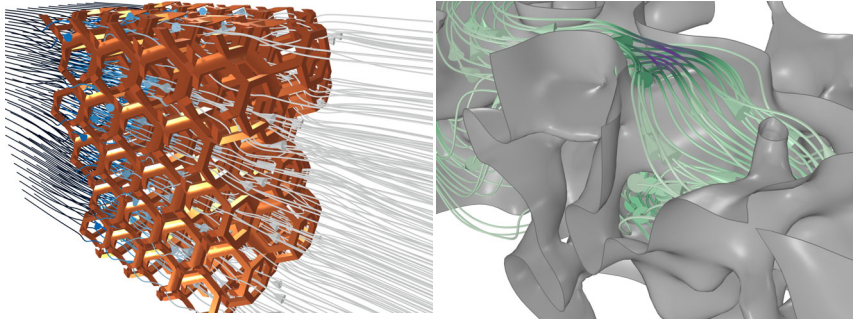


Figure 2-1: Flow through porous structures.

If it is possible to calculate averaged values from the simulation on a microscopic scale (Figure 2-1), which are also representative for any other section, the modeled section is called a representative elementary volume (REV). Two variables mainly characterize the porous material: The porosity describes the ratio of void/pore volume to total volume

$$\epsilon_p = \frac{V_v}{V_{\text{tot}}}$$

and the permeability $\kappa(\text{m}^2)$ specifies the ability of a fluid to pass through it.

About Darcian and Non-Darcian Flow

The basic law that describes the flow in porous materials is Darcy's law. It describes a linear relationship between the velocity field \mathbf{u} (m/s) and the gradient of the pressure p (Pa).

$$\mathbf{u} = -\frac{\kappa}{\mu} \nabla p \quad (2-1)$$

This is only valid for very low velocities or at low Reynolds numbers ($\text{Re} < 10$)



See the section [The Reynolds Number](#) in the *COMSOL Multiphysics Reference Manual*.

In case of relatively fast flow ($\text{Re} > 10$) or relatively high Knudsen numbers ($\text{Kn} > 0.1$) Darcy's linear relation between velocity and pressure drop is no longer valid. Therefore, different [Permeability Models](#) have been introduced to capture these effects. A general form of the nonlinear relationship of pressure gradient to velocity can be written as

$$-\nabla p = \frac{\mu}{\kappa} \mathbf{u} + \beta \rho \mathbf{u} |\mathbf{u}| \quad (2-2)$$

The flow regime through a packed bed may be identified from the Reynolds number of the bed. According to textbooks, for Reynolds numbers $\text{Re} < 10$ the flow can be described by [Kozeny–Carman](#) equation (Darcian flow). For $10 < \text{Re} < 1000$, which is sometimes called the *transitional regime*, the flow is better described by the non-Darcian [Ergun](#) equation, and for $\text{Re} > 1000$ Ergun's equation can be approximated by the [Burke-Plummer](#) equation for turbulent flows.

Permeability Models

Different models can define the capacity of the porous medium to transmit flow. Some use the permeability, κ (SI unit: m^2), and the viscosity of the fluid, μ (SI unit: $\text{Pa}\cdot\text{s}$); others use the hydraulic conductivity, K (SI unit: m/s). Several relationships have been established to describe Darcian and non-Darcian flow through different porous materials.

PERMEABILITY

The capacity of the porous medium to transmit flow cannot be described by porosity alone, as the shape and orientation of the pores is also an important factor. This is described by the permeability κ . For an isotropic medium κ is a scalar quantity. Often porous materials are anisotropic and κ then is a tensor.

There is also the term relative permeability, which is relevant for multiphase flow in porous media applications.



For modeling multiphase flow in porous media [The Multiphase Flow in Porous Media Interface](#) is available. For flow through unsaturated porous media, also [The Richards' Equation Interface](#) is available.

HYDRAULIC CONDUCTIVITY

The hydraulic conductivity represents both fluid and solid properties. If the model is defined using the hydraulic conductivity as in:

$$\frac{\kappa}{\mu} = \frac{K}{\rho g}$$

[Equation 2-1](#) changes to

$$\mathbf{u} = -\frac{\mathbf{K}}{\rho g} \nabla p \quad (2-3)$$

KOZENY-CARMAN

The Kozeny–Carman equation describes the flow through granular soils and packed beds by estimating the permeability of the porous medium from the porosity ϵ_p and average particle diameter d_p (see, for example, [Ref. 5](#)).

$$\kappa = \frac{d_p^2}{180} \frac{\epsilon_p^3}{(1 - \epsilon_p)^2}$$

ERGUN

The Ergun equation is an extension of Kozeny–Carman equation to higher Reynolds numbers ($Re > 10$), where Darcy's linear relation between pressure drop and velocity is no longer valid. In Ergun equation the linear relation between pressure drop and velocity is augmented by a quadratic term. The pressure drop ([Ref. 6](#)) in Ergun's equation is given by:

$$-\nabla p = \frac{150(1 - \epsilon_p)^2}{d_p^2 \epsilon_p^3} \mu \mathbf{u} + \frac{1.75(1 - \epsilon_p)}{d_p \epsilon_p^3} \rho \mathbf{u} |\mathbf{u}|$$

Here, d_p is the effective (average) particle diameter in the porous medium and ϵ_p is the porosity. With definitions of permeability κ and parameter β (SI unit 1/m) as

$$\kappa = \frac{d_p^2}{150} \frac{\epsilon_p^3}{(1 - \epsilon_p)^2} \quad (2-4)$$

and

$$\beta = \frac{1.75(1 - \epsilon_p)}{d_p \epsilon_p^3} \quad (2-5)$$

we get [Equation 2-2](#) and Ergun's velocity can then be calculated as

$$\mathbf{u} = -\frac{\kappa}{\mu} \frac{2}{1 + \sqrt{1 + 4\left(\frac{\kappa}{\mu}\right)^2 \beta \rho |\nabla p|}} \nabla p$$

Using the Reynolds number Re and the friction factor f_p for packed beds

$$Re = \frac{d_p \rho |\mathbf{u}|}{(1 - \epsilon_p) \mu}$$

$$f_p = |\nabla p| \frac{d_p}{\rho \mathbf{u}^2} \frac{\epsilon_p^3}{2(1 - \epsilon_p)} = \frac{150}{Re} + 1.75,$$

Ergun's velocity can alternatively be written as

$$\mathbf{u} = -\frac{\kappa}{\mu} \left(1 - \frac{1.75}{f_p}\right) \nabla p$$

From this expression, we observe that for low Reynolds numbers $\text{Re} < 1$ (or equivalently, high friction factors) the flow can be described by the linear Darcy's Law (Equation 2-1).

FORCHHEIMER

For increasing filter velocities ($\text{Re} > 10$), inertial effects and turbulent friction forces become significant. In the Forchheimer drag this is considered by adding an additional term to Darcy's law:

$$-\nabla p = \frac{\mu}{\kappa} \mathbf{u} + \frac{c_F}{\sqrt{\kappa}} \rho \mathbf{u} |\mathbf{u}| \quad (2-6)$$

Here, the Forchheimer parameter c_F is dimensionless. The permeability κ is specified by the user and is not computed from porosity nor particle diameter as for Ergun's equation. If the parameter β is defined as

$$\beta = \frac{c_F}{\sqrt{\kappa}}$$

Equation 2-6 becomes equivalent to Equation 2-1.

BURKE-PLUMMER

The Burke-Plummer equation is applicable in flow regimes with Reynolds number $\text{Re} > 1000$. Here, the pressure drop is proportional to the square of the velocity:

$$\frac{\Delta p}{L} = \frac{1.75(1 - \epsilon_p)}{d_p^3 \epsilon_p^3} \rho v^2$$

here, L is the length of the packed bed, Δp is the pressure drop d_p is the effective (average) particle diameter in the porous medium, ϵ_p is the porosity, and v is the velocity magnitude.

Using the same definition for the parameter β (SI unit $1/\text{m}$) as in Ergun's equation (Equation 2-5) the pressure drop is related to the velocity as

$$\frac{\Delta p}{L} = \beta \rho v^2$$

which means that the velocity magnitude can be written in terms of the pressure gradient as

$$v = \sqrt{\frac{|\nabla p|}{\beta\rho}}$$

As flow travels from areas of higher pressure to areas of lower pressure, [Equation 2-2](#) can be more generally written as

$$-\nabla p = \beta\rho|\mathbf{u}|\mathbf{u}$$

replacing

$$|\mathbf{u}| = v = \sqrt{|\nabla p|/\beta\rho}$$

we get an expression for the velocity field in terms of the pressure gradient

$$\mathbf{u} = -\frac{1}{\sqrt{\beta\rho|\nabla p|}}\nabla p$$

Note that the viscosity is not used to define the velocity as in Darcy's law, but only the fluid density, the porosity and particle diameter. Note also that the analytical friction factor for Burke-Plummer equation is $f_p = 1.75$. A small parameter is added inside the square-root to avoid division by zero when the pressure gradient approaches zero.

KLINKENBERG

The Klinkenberg effect becomes important for gas flows in porous media where the mean free path of the gas molecules is comparable to the pore dimensions. In this case, molecular collisions with the pore walls occur more often than collisions between molecules. To take this effect into account, Klinkenberg ([Ref. 7](#)) derived the following expression for the permeability:

$$\kappa = \kappa_{\infty} \left(1 + \frac{b_K}{p_A} \right) \quad (2-7)$$

here, κ_{∞} is the permeability of the gas at high pressure and density (Knudsen number $0.001 < \text{Kn} < 0.1$), b_K is the Klinkenberg parameter (also called *Klinkenberg slip factor*, or *gas slippage factor*), whose default value is set to 1 kPa according to [Ref. 8](#), and p_A is the absolute pressure as defined in the Darcy's Law interface ($p_A = p + p_{\text{ref}}$).

Mass and Momentum Conservation in Porous Media

In this chapter:

- [Mass Conservation](#)
- [Momentum Conservation](#)

Mass Conservation

In a volume of pure fluid with density ρ and viscosity μ the mass conservation is defined by the continuity equation

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{v}) = 0 \quad (2-8)$$

with the fluid velocity \mathbf{v} . For an incompressible fluid it simplifies to:

$$\rho \nabla \cdot \mathbf{v} = 0$$

For a porous medium [Equation 2-8](#) must be corrected by the porosity ϵ_p because the fluid volume is only a fraction of the total volume, hence

$$\frac{\partial \epsilon_p \rho}{\partial t} + \nabla \cdot (\epsilon_p \rho \mathbf{v}) = 0 \quad (2-9)$$

Note that \mathbf{v} refers to the velocity of the fluid within the pores also called seepage or interstitial velocity. More commonly used to describe the flow through a porous medium is the superficial velocity or Darcy velocity u :

$$u = \frac{Q}{A} \quad (2-10)$$

Q (SI unit: m³/s) is the volume flow rate of the fluid and A (SI unit: m²) is the cross sectional area of the porous medium. The cross-sectional area for the pore spaces which is available for the fluid to flow is related to the porosity by $A_e = \epsilon_p A$ such that the Darcy velocity and seepage velocity relate as follows:

$$\mathbf{u} = \epsilon_p \mathbf{v} \quad (2-11)$$

And [Equation 2-9](#) can be written as:

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

or when a volumetric mass source Q_m (SI unit: kg/m³/s) is present:

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

Note that Q_m is a mass source per unit volume of the porous medium (not per unit pore volume).

Momentum Conservation

There are numerous ways for deriving the momentum equation in porous media. Here, start from the assumption that the pressure drop in a fluid results from viscous losses only and transient and inertial effects are neglected:

$$\nabla p = \mathbf{f}_{\text{viscous}}$$

The pressure p is the fluid pore pressure and for low velocities the viscous force depends linearly on the velocity ([Equation 2-10](#)) according to $\nabla p \sim \mathbf{u}$. This relation is well known as Darcy's law ([Equation 2-1](#)). The situation becomes more complicated when transient and inertial effects must be considered. For a pure fluid the governing equations are the Navier Stokes equations (see the section [General Single-Phase Flow Theory](#) in the *COMSOL Multiphysics Reference Manual*). The momentum conservation ([Equation 1-2](#)) rewritten for the fluid in a porous medium reads:

$$\epsilon_p \rho \left(\frac{\partial \mathbf{v}}{\partial t} + (\mathbf{v} \cdot \nabla) \mathbf{v} \right) = \epsilon_p (\nabla \cdot [-p \mathbf{I} + \mathbf{K}]) + \epsilon_p \mathbf{f}_{\text{viscous}} \quad (2-12)$$

Remember that \mathbf{v} denotes the interstitial velocity and \mathbf{K} is the stress tensor. [Equation 2-12](#) describes the momentum conservation per unit volume of the porous medium. It is more common to formulate the equation in terms of Darcy velocity. Dividing by ϵ_p and using [Equation 2-11](#) yields:

$$\frac{\rho}{\epsilon_p} \left(\frac{\partial \mathbf{u}}{\partial t} + \frac{1}{\epsilon_p} (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = \nabla \cdot [-p \mathbf{I} + \mathbf{K}] + \mathbf{f}_{\text{viscous}}$$

The pressure drop due to viscous losses is described in [Equation 2-2](#) and one obtains

$$\frac{\rho}{\epsilon_p} \left(\frac{\partial \mathbf{u}}{\partial t} + \frac{1}{\epsilon_p} (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = \nabla \cdot [-p \mathbf{I} + \mathbf{K}] + (\mu \kappa^{-1} + \beta \rho |\mathbf{u}|) \mathbf{u} \quad (2-13)$$

A mass source or sink term adds a momentum source/sink as well and adds the term $-\varepsilon_p Q \mathbf{v}$ to the right hand side of Equation 2-12. In terms of Darcy velocity Equation 2-13 can be rewritten as


$$\frac{\rho}{\varepsilon_p} \left(\frac{\partial \mathbf{u}}{\partial t} + \frac{1}{\varepsilon_p} (\mathbf{u} \cdot \nabla) \mathbf{u} \right) = \nabla \cdot [-p \mathbf{I} + K] + \left(\mu \kappa^{-1} + \beta \rho |\mathbf{u}| - \frac{Q}{\varepsilon_p} \right) \mathbf{u}$$

with


$$Q = \frac{Q_m}{\varepsilon_p}$$

Thus it results in the Brinkman equations (Equation 4-26) used by the The Brinkman Equations Interface.

Single- and Multiphase Flow Interfaces

This chapter describes the interfaces found under the **Fluid Flow>Single-Phase Flow** branch (

- [The Laminar Flow and Creeping Flow Interfaces](#)
- [Theory for the Single-Phase Flow Interfaces.](#)

Under the **Fluid Flow>Multiphase Flow** branch () the following interfaces are available:

- The Phase Transport Interface
- The Phase Transport in Porous Media Interface

They are fully described in Section [The Phase Transport Interfaces](#) of the *CFD Module User's Guide*.


The Laminar Flow and Creeping Flow Interfaces

In this section:


- [The Creeping Flow Interface](#)
- [The Laminar Flow Interface](#)
- [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#)



- [Theory for the Single-Phase Flow Interfaces](#)
-

In the *COMSOL Multiphysics Reference Manual*, see [Table 2-4](#) for links to common sections such as **Discretization**, **Consistent Stabilization**, **Inconsistent Stabilization**, and **Advanced Settings** sections, some of them accessed by clicking the **Show** button () and choosing the applicable option. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

The Creeping Flow Interface

The **Creeping Flow (spf)** interface () is used for simulating fluid flows at very low Reynolds numbers for which the inertial term in the Navier–Stokes equations can be neglected. Creeping flow, also referred to as Stokes flow, occurs in systems with high viscosity or small geometrical length scales (for example, in microfluidics and MEMS devices). The fluid can be compressible or incompressible, as well as Newtonian or non-Newtonian.

The equations solved by the Creeping Flow interface are the Stokes equations for conservation of momentum and the continuity equation for conservation of mass.

The Creeping Flow interface can be used for stationary and time-dependent analyses.

The main feature is the **Fluid Properties** node, which adds the Stokes equations and provides an interface for defining the fluid material and its properties. Except where noted below, see [The Laminar Flow Interface](#) for all other settings.

PHYSICAL MODEL


By default, the **Neglect inertial term (Stokes flow)** check box is selected. If unchecked, the inertial terms are included in the computations.

DISCRETIZATION

By default, the Creeping Flow interface uses **P2+P1** elements. Contrary to general laminar and turbulent single-phase flow simulations employing purely linear P1+P1 elements, P2+P1 elements are well suited for most creeping flow simulations but are not in general fully conservative. Full conservation properties and high accuracy is obtained by selecting P2+P2 or P3+P3 elements.

Note that P1+P1, P2+P2, and P3+P3 discretizations (the so-called equal order interpolations) require streamline diffusion to be active.

The Laminar Flow Interface

The **Laminar Flow (spf)** interface () is used to compute the velocity and pressure fields for the flow of a single-phase fluid in the laminar flow regime. A flow remains laminar as long as the Reynolds number is below a certain critical value. At higher Reynolds numbers, disturbances have a tendency to grow and cause transition to turbulence. This critical Reynolds number depends on the model, but a classical example is pipe flow, where the critical Reynolds number is known to be approximately 2000.

The physics interface supports incompressible flow, weakly compressible flow (the density depends on temperature but not on pressure), and compressible flow at low Mach numbers (typically less than 0.3). It also supports flow of non-Newtonian fluids.

The equations solved by the Laminar Flow interface are the Navier–Stokes equations for conservation of momentum and the continuity equation for conservation of mass.

The Laminar Flow interface can be used for stationary and time-dependent analyses. Time-dependent studies should be used in the high-Reynolds number regime as these flows tend to become inherently unsteady.

When the Laminar Flow interface is added, the following default nodes are also added in the **Model Builder: Fluid Properties, Wall** (the default boundary condition is **No slip**), and **Initial Values**. Other nodes that implement, for example, boundary conditions and volume forces, can be added from the **Physics** toolbar or from the context menu displayed when right-clicking **Laminar Flow**.

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. Physics interface variables can be referred to 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 `spf`.

About the Physics Interface Label Names

The label for a single-phase flow interface (the node name in the Model Builder) is dynamic and is reset according to the turbulence model and the neglect inertial term (Stokes flow) property set at the physics interface level. The availability also depends on the software license.

If the **Neglect inertial term (Stokes flow)** check box is selected, then the Label changes to **Creeping Flow**, which is the same Label that displays when that interface is added from the Model Wizard or Add Physics window.

PHYSICAL MODEL

Compressibility

Depending of the fluid properties and the flow regime, three options are available for the **Compressibility** option. In general the computational complexity increases from **Incompressible flow** to **Weakly compressible flow** to **Compressible flow (Ma<0.3)** but the underlying hypotheses are increasingly more restrictive in the opposite direction.

When the **Incompressible flow** option (default) is selected, the incompressible form of the Navier–Stokes and continuity equations is applied. In addition, the fluid density is evaluated at the **Reference pressure level** defined in this section. The **Reference temperature** is set to 293.15 K.

The **Weakly compressible flow** option models compressible flow when the pressure dependency of the density can be neglected. When selected, the compressible form of the Navier–Stokes and continuity equations is applied. In addition, the fluid density is evaluated at the **Reference pressure level** defined in this section.

When the **Compressible flow (Ma<0.3)** option is selected, the compressible form of the Navier–Stokes and continuity equations is applied. $Ma < 0.3$ indicates that the inlet and outlet conditions, as well as the stabilization, may not be suitable for transonic and supersonic flow. For more information, see [The Mach Number Limit](#).

Neglect Inertial Term (Stokes Flow)

For low Reynolds number flows, the inertial terms in the Navier–Stokes equations may be neglected.

Porous Media Domains

With the addition of various modules, the **Enable porous media domains** check box is available. Selecting this option, a **Fluid and Matrix Properties** node, a **Mass Source** node, and a **Forchheimer Drag** subnode are added to the physics interface. These are described for the **Brinkman Equations** interface in the respective module’s documentation. The **Fluid and Matrix Properties** can be applied on all domains or on a subset of the domains.

Include Gravity

When the **Include gravity** check box is selected, a global **Gravity** feature is shown in the interface model tree, and the buoyancy force is included in the Navier–Stokes equations.

Also, when the **Include gravity** check box is selected, the **Use reduced pressure** option changes the pressure formulation from using the total pressure (default) to using the reduced pressure. This option is suitable for configurations where the density changes are very small; otherwise, the default formulation can be used. For more information, see [Gravity](#)

Reference Values

Reference values are global quantities used to evaluate the density of the fluid when the **Incompressible flow** or the **Weakly compressible flow** option is selected.

Reference pressure level There are generally two ways to include the pressure in fluid flow computations: either to use the absolute pressure $p_A = p + p_{\text{ref}}$, or the gauge pressure p . When p_{ref} is nonzero, the physics interface solves for the gauge pressure whereas material properties are evaluated using the absolute pressure. The reference pressure level is also used to define the reference density.

Reference temperature The reference temperature is used to define the reference density.

Reference position When **Include gravity** is selected, the reference position can be defined. It corresponds to the location where the total pressure (that includes the hydrostatic pressure) is equal to the **Reference pressure level**.

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface — the **Velocity field \mathbf{u}** and its components, and the **Pressure p** .

If required, the names of the field, component, and dependent variable can be edited. 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. Component names must be unique within a model except when two fields share a common field name.

CONSISTENT STABILIZATION

To enable this section, click the **Show More Options** button () and select **Stabilization** in the **Show More Options** dialog box.

There are two *consistent stabilization methods*: **Streamline diffusion** and **Crosswind diffusion**. Usually, both check boxes for these methods are selected by default and should remain selected for optimal performance. Consistent stabilization methods do not perturb the original transport equation. Streamline diffusion must be selected when using equal-order interpolation for pressure and velocity.

Select the **Use dynamic subgrid time scale** check box to approximate the time-scale tensor in time dependent problems from projections of weak expressions. This check box is selected by default. When not selected the actual time-step is used.

INCONSISTENT STABILIZATION

To enable this section, click the **Show More Options** button () and select **Stabilization** in the **Show More Options** dialog box.

There is usually just one *inconsistent stabilization method* — **Isotropic diffusion**. This method is equivalent to adding a term to the diffusion coefficient in order to dampen the effect of oscillations by making the system somewhat less dominated by convection. If possible, minimize the use of the inconsistent stabilization method because by using it you no longer solve the original problem. By default, the **Isotropic diffusion** check box is not selected because this type of stabilization adds artificial diffusion and affects the accuracy of the original problem. However, this option can be used to get a good initial guess for underresolved problems.


If required, select the **Isotropic diffusion** check box and enter a **Tuning parameter δ_{id}** as a scalar positive value. The default value is 0.25 (a reasonable value to start with is

roughly 0.5 divided by the element order). A higher value adds more isotropic diffusion.



- [Numerical Stability — Stabilization Techniques for Fluid Flow](#)

ADVANCED SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box. Normally these settings do not need to be changed.

The **Use pseudo time stepping for stationary equation form** is per default set to **Automatic from physics**. This option can add pseudo time derivatives to the equation when the **Stationary equation** form is used in order to speed up convergence. Pseudo time stepping is triggered when the Laminar Flow interface is selected in some multiphysics coupling features. Set **Automatic from physics** to **On** to apply pseudo time stepping also for laminar flows. Set it to **Off** to disable pseudo time stepping completely.

When **Use pseudo time stepping for stationary equation form** is set to **Automatic from physics** or **On**, a **CFL number expression** should also be defined. For the default **Automatic** option, the local CFL number (from the Courant–Friedrichs–Lewy condition) is determined by a PID regulator.

The **Use Block Navier-Stokes preconditioner in time dependent studies** check box under **Linear solvers** is available when the **Compressibility** option is set to **Incompressible flow**. When this check box is selected, the default solver for time dependent study steps will use the Block Navier-Stokes preconditioner in iterative solvers for the velocity and pressure. Using this preconditioner may result in shorter solution times for large time dependent problems with high Reynolds numbers.



- [Pseudo Time Stepping for Laminar Flow Models](#)
- [Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow](#)
- [Theory for the Single-Phase Flow Interfaces](#)
- [Block Navier–Stokes](#) in the *COMSOL Multiphysics Reference Manual*.

DISCRETIZATION

The default discretization for Laminar Flow is **P1+P1** elements — that is, piecewise linear interpolation for velocity and pressure. This is suitable for most flow problems.

Some higher-order interpolations are also available, and they can be cost-effective options to obtain high accuracy for flows with low Reynolds numbers.

The **P2+P2** and **P3+P3** options, the equal-order interpolation options, are the preferred higher-order options because they have higher numerical accuracy than the mixed-order options **P2+P1** and **P3+P2**. The equal-order interpolation options do, however, require streamline diffusion to be active.



Flow Past a Cylinder: Application Library path
COMSOL_Multiphysics/Fluid_Dynamics/cylinder_flow

Domain, Boundary, Pair, and Point Nodes for Single-Phase Flow

The following nodes, listed in alphabetical order, are available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking to access the context menu (all users).

- Boundary Stress
- Flow Continuity
- Fluid Properties
- Gravity
- Initial Values
- Inlet
- Line Mass Source¹
- Open Boundary
- Outlet
- Periodic Flow Condition
- Point Mass Source¹
- Pressure Point Constraint
- Symmetry
- Volume Force
- Wall

¹ A feature that may require an additional license



For 2D axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and adds an **Axial Symmetry** node that is valid on the axial symmetry boundaries only.

In the *COMSOL Multiphysics Reference Manual*, see [Table 2-4](#) for links to common sections and [Table 2-5](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Fluid Properties

The **Fluid Properties** node adds the momentum and continuity equations solved by the physics interface, except for volume forces, which are added by the [Volume Force](#) feature. The node also provides an interface for defining the material properties of the fluid.

MODEL INPUTS

Fluid properties, such as density and viscosity, can be defined through user inputs, variables, or by selecting a material. For the latter option, additional inputs, for example temperature or pressure, may be required to define these properties.

Temperature

By default, the **Temperature** model input is set to **Common model input**, and the temperature is controlled from [Default Model Inputs](#) under **Global Definitions** or by a locally defined [Model Input](#). If a Heat Transfer interface is included in the component, it controls the temperature **Common model input**. Alternatively, the temperature field can be selected from another physics interface. All physics interfaces have their own tags (**Name**). For example, if a Heat Transfer in Fluids interface is included in the component, the **Temperature (ht)** option is available for T .


You can also select **User defined** from the **Temperature** model input in order to manually prescribe T .

Absolute Pressure

This input appears when a material requires the absolute pressure as a model input. The absolute pressure is used to evaluate material properties, but it also relates to the value of the calculated pressure field. There are generally two ways to calculate the pressure when describing fluid flow: either to solve for the absolute pressure or for a pressure (often denoted gauge pressure) that relates to the absolute pressure through a reference pressure.

The choice of pressure variable depends on the system of equations being solved. For example, in a unidirectional incompressible flow problem, the pressure drop over the modeled domain is probably many orders of magnitude smaller than the atmospheric pressure, which, when included, may reduce the stability and convergence properties of the solver. In other cases, such as when the pressure is part of an expression for the gas volume or the diffusion coefficients, it may be more convenient to solve for the absolute pressure.

The default **Absolute pressure** p_A is $p + p_{\text{ref}}$, where p is the dependent pressure variable from the Navier–Stokes or RANS equations, and p_{ref} is from the user input defined at the physics interface level. When p_{ref} is nonzero, the physics interface solves for a gauge pressure. If the pressure field instead is an absolute pressure field, p_{ref} should be set to 0.

The **Absolute pressure** field can be edited by clicking **Make All Model Inputs Editable** () and entering the desired value in the input field.



FLUID PROPERTIES

Density

The density can either be specified by a material, or by a **User defined** expression. The density in a material can depend on temperature and pressure, and these dependencies are automatically replaced by p_{ref} for weakly compressible flows and p_{ref} and T_{ref} for incompressible flows (as specified by the **Compressibility** setting at the physics interface level). If density variations with respect to pressure are to be included in the computations, **Compressibility** must be set to compressible. Any dependencies in the density on quantities other than temperature and pressure must be consistent with the **Compressibility** setting at the interface level.

Dynamic Viscosity

The **Dynamic viscosity** μ describes the relationship between the shear rate and the shear stresses in a fluid. Intuitively, water and air have low viscosities, and substances often described as thick (such as oil) have higher viscosities.

Volume Force

The **Volume Force** node specifies the volume force \mathbf{F} on the right-hand side of the momentum equation.

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot \left[-p \mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu(\nabla \cdot \mathbf{u}) \mathbf{I} \right] + \mathbf{F}$$

If several volume-force nodes are added to the same domain, then the sum of all contributions are added to the momentum equation.

Initial Values

The initial values serve as initial conditions for a transient simulation or as an initial guess for a nonlinear solver in a stationary simulation. Note that for a transient compressible-flow simulation employing a material for which the density depends on the pressure (such as air), discontinuities in the initial values trigger pressure waves even when the Mach number is small. The pressure waves must be resolved and this puts a restriction on the time step.

INITIAL VALUES

Initial values or expressions should be specified for the **Velocity field** \mathbf{u} and the **Pressure** p .

Wall

The **Wall** node includes a set of boundary conditions describing fluid-flow conditions at stationary, moving, and leaking walls. For turbulent flow, the description may involve wall functions and asymptotic expressions for certain turbulence variables.

BOUNDARY CONDITION

Select a **Boundary condition** for the wall.

- [No Slip](#)
- [Slip](#)
- [Leaking Wall](#)

No Slip

No slip is the default boundary condition to model solid walls. A no slip wall is a wall where the fluid velocity relative to the wall velocity is zero. For a stationary wall that means that $\mathbf{u} = 0$.

The option for **Porous treatment of no slip condition** should be decided in **Physical Model** when **Enable porous media domains** is activated. It specifies how **Wall** boundaries and **Interior Wall** boundaries internal to porous domains are treated. When **Standard no slip formulation** is chosen, a common no slip condition is applied on all solid walls. When **Porous slip** is chosen, a blending analytic expression is instead applied on the corresponding wall boundaries adjacent to the porous medium domain. It results in a no slip condition in case the porous length scale is fully resolved by the mesh and a slip condition in the opposite limit when the mesh is much coarser than the porous scale. A smooth transition between these limits is ensured. Note that the interpretation and usage of non-zero slip at the wall is the same as in the **Navier Slip** boundary condition. By default, **Velocity formulation** is on and the treatment is based on an approximate reconstruction of the far field pressure gradient using the slip velocity at the wall. If the **Pressure-gradient formulation** is chosen, the local pressure gradient at the wall is employed.

Slip

The **Slip** option prescribes a no-penetration condition, $\mathbf{u} \cdot \mathbf{n} = 0$. It is implicitly assumed that there are no viscous effects at the slip wall and hence, no boundary layer develops. From a modeling point of view, this can be a reasonable approximation if the main effect of the wall is to prevent fluid from leaving the domain.

Leaking Wall

This boundary condition may be used to simulate a wall where fluid is leaking into or leaving the domain with the velocity $\mathbf{u} = \mathbf{u}_1$ through a perforated wall. The components of the **Fluid velocity** \mathbf{u}_1 on the leaking wall should be specified.

WALL MOVEMENT

This section contains controls to describe the wall movement relative to the lab (or spatial) frame.

The **Translational velocity** setting controls the translational wall velocity, \mathbf{u}_{tr} . The list is per default set to **Automatic from frame**. The physics automatically detects if the spatial frame moves. This can for example happen if an ALE interface is present in the model component. If there is no movement $\mathbf{u}_{tr} = \mathbf{0}$. If the frame moves, \mathbf{u}_{tr} becomes equal to the frame movement. \mathbf{u}_{tr} is accounted for in the actual boundary condition prescribed in the **Boundary condition** section.


Select **Zero (Fixed wall)** from **Translational velocity** selection list to prescribe $\mathbf{u}_{tr} = \mathbf{0}$.

Select **Manual** from **Translational velocity** selection list in order to manually prescribe **Velocity of moving wall**, \mathbf{u}_{tr} . This can for example be used to model an oscillating wall where the magnitude of the oscillations are very small compared to the rest of the model. Specifying translational velocity manually does not automatically cause the associated wall to move. An additional Moving Mesh node needs to be added from Definitions to physically track the wall movement in the spatial reference frame.

The **Sliding wall** option is appropriate if the wall behaves like a conveyor belt with the surface sliding in a tangential direction. A velocity is prescribed at the wall and the boundary itself does not have to actually move relative to the reference frame.

- For 3D components, values or expressions for the **Velocity of sliding wall** \mathbf{u}_w should be specified. If the velocity vector entered is not in the plane of the wall, COMSOL Multiphysics projects it onto the tangential direction. Its magnitude is adjusted to be the same as the magnitude of the vector entered.
- For 2D components, the tangential direction is unambiguously defined by the direction of the boundary. For this reason, the sliding wall boundary condition has different definitions in different space dimensions. A single entry for the **Velocity of the tangentially moving wall** U_w should be specified in 2D.

CONSTRAINT SETTINGS

This section is displayed by clicking the **Show More Options** button () and selecting **Advanced Physics Options** in the **Show More Options** dialog box. The **Constraints** settings

can be set to **Default**, **Use pointwise constraints**, **Use DG constraints**, or **Use weak constraints**. **Use mixed constraints** can be selected when imposing a no slip condition exactly.

Depending on the constraint method selected, the following settings are available:

- **Apply reaction terms on** can be set to **Individual dependent variables** (default) or **All physics (symmetric)**. This setting is not available when **Use DG constraints** is selected.
- Select **Elemental** (default) or **Nodal** under **Constraint method**. This setting is not available for **Use DG constraints** or **Use weak constraints**.



- [Theory for the Wall Boundary Condition](#)
- [The Moving Mesh Interface](#) in the *COMSOL Multiphysics Reference Manual*

Inlet

This condition should be used on boundaries for which there is a net flow into the domain. To obtain a numerically well-posed problem, it is advisable to also consider the Outlet conditions when specifying an Inlet condition. For example, if the pressure is specified at the outlet, the velocity may be specified at the inlet, and vice versa. Specifying the velocity vector at both the inlet and the outlet may cause convergence difficulties.

BOUNDARY CONDITION

The available **Boundary condition** options for an inlet are **Velocity**, **Fully developed flow**, and **Pressure**. After selecting a **Boundary Condition** from the list, a section with the same or a similar name displays underneath. For example, if **Velocity** is selected, a **Velocity** section, where further settings are defined, is displayed.

VELOCITY

The **Normal inflow velocity** is specified as $\mathbf{u} = -\mathbf{n}U_0$, where \mathbf{n} is the boundary normal pointing out of the domain and U_0 is the normal inflow speed.

The **Velocity field** option sets the velocity vector to $\mathbf{u} = \mathbf{u}_0$. The components of the inlet velocity vector \mathbf{u}_0 should be defined for this choice.

PRESSURE CONDITIONS

This option specifies the normal stress, which in most cases is approximately equal to the pressure.

- For incompressible flow, the **Pressure** list has two options, **Static** and **Total**. For weakly compressible and compressible flow, the static pressure should be specified in the text field.
 - If **Pressure** is **Static**, and the reference pressure p_{ref} , defined at the physics interface level, is equal to 0, the value of the pressure p_0 , at the boundary, is the absolute pressure. Otherwise, p_0 is the relative pressure at the boundary.
 - If **Pressure** is **Total**, the **Average** check box is available and unselected by default to prescribe the total pressure pointwise. If it is selected, the averaged total pressure is imposed in the weak forms instead.

When **Include gravity** is selected and **Use reduced pressure** not selected in the interface **Physical model** section, the **Compensate for hydrostatic pressure approximation** (named **Compensate for hydrostatic pressure** for incompressible flows) check box is available and selected by default. When it is selected, the hydrostatic pressure is automatically added to the pressure entered in p_0 user input.

- The **Suppress backflow** option adjusts the inlet pressure locally in order to reduce the amount of fluid exiting the domain through the boundary. If you clear the suppress backflow option, the inlet boundary can become an outlet depending on the pressure field in the rest of the domain.
- **Flow direction** controls in which direction the fluid enters the domain.
 - For **Normal flow**, it prescribes zero tangential velocity component.
 - For **User defined**, an **Inflow velocity direction** \mathbf{d}_u (dimensionless) should be specified. The magnitude of \mathbf{d}_u does not matter, only the direction. \mathbf{d}_u must point into the domain.

FULLY DEVELOPED FLOW

The **Fully developed flow** option adds contributions to the inflow boundary, which force the flow toward the solution for a fully developed channel flow. The channel can be thought of as a virtual extrusion of the inlet cross section. The inlet boundary must hence be flat in order for the fully developed flow condition to work properly. In 2D axisymmetric models, the inlet normal must be parallel to the symmetry axis.

Select an option to control the flow rate at the inlet:

- **Average velocity**, U_{av} .


- **Flow rate**, V_0 . Two-dimensional models also require an **Entrance thickness**, D_z , which is the out-of-plane thickness of the extruded entrance channel.
- **Average pressure**, P_{av} . Note that P_{av} is the average pressure on the inflow boundary.



The **Apply condition on each disjoint selection separately** check box is selected per default. When this setting is selected, the fully developed flow condition is applied separately on each disjoint selection. If this option is not selected, the condition is applied over the whole feature selection. The **Apply condition on each disjoint selection separately** should be disabled only if the flow conditions are known to be identical on each disjoint boundary.

The fully developed flow condition requires any volume force to be approximately aligned with the normal of the inlet boundary. The exception is gravity when the **Include gravity** setting is selected in the physics interface settings. Unless **Use reduced pressure** is also selected, an option to **Compensate for hydrostatic pressure** or **Compensate for hydrostatic pressure approximation** becomes available. It is selected per default and should only be deselected if the inlet normal is aligned with the gravity force and you want to specify an average pressure that includes the hydrostatic pressure.

A fully developed flow boundary cannot be adjacent to any Interior Wall feature.

CONSTRAINT SETTINGS

This section is displayed by clicking the **Show** button () and selecting **Advanced Physics Options**.

	<ul style="list-style-type: none"> • Prescribing Inlet and Outlet Conditions • Normal Stress Boundary Condition
	Fully Developed Flow (Inlet)

Outlet

This condition should be used on boundaries for which there is a net outflow from the domain. To obtain a numerically well-posed problem, it is advisable to also consider the Inlet conditions when specifying an Outlet condition. For example, if the velocity is specified at the inlet, the pressure may be specified at the outlet, and vice versa.

Specifying the velocity vector at both the inlet and the outlet may cause convergence difficulties. Selecting appropriate outlet conditions for the Navier–Stokes equations is a nontrivial task. Generally, if there is something interesting happening at an outflow boundary, the computational domain should be extended to include this phenomenon.

BOUNDARY CONDITION

The available **Boundary condition** options for an outlet are **Pressure**, **Fully developed flow**, and **Velocity**.

PRESSURE CONDITIONS

This option specifies the normal stress, which in most cases is approximately equal to the pressure. The tangential stress component is set to zero.

- For incompressible flow, the **Pressure** list has two options, **Static** and **Total**. For weakly compressible and compressible flow, the static pressure should be specified in the text field.
 - If **Pressure** is **Static**, and the reference pressure p_{ref} , defined at the physics interface level, is equal to 0, the value of the pressure p_0 , at the boundary, is the absolute pressure. Otherwise, p_0 is the relative pressure at the boundary.
 - If **Pressure** is **Total**, the **Average** check box is available and selected by default to prescribe the averaged total pressure in the weak forms. If it is unselected, the total pressure is imposed pointwise.

When **Include gravity** is selected and **Use reduced pressure** not selected in the interface **Physical model** section, the **Compensate for hydrostatic pressure approximation** (named **Compensate for hydrostatic pressure** for incompressible flows) check box is available and selected by default. When it is selected, the hydrostatic pressure is automatically added to the pressure entered in p_0 user input.

- The **Normal flow** option changes the no tangential stress condition to a no tangential velocity condition. This forces the flow to exit (or enter) the domain perpendicularly to the outlet boundary.
- The **Suppress backflow** check box is selected by default. This option adjusts the outlet pressure in order to reduce the amount of fluid entering the domain through the boundary.

VELOCITY

See the **Inlet** node **Velocity** section for the settings.

FULLY DEVELOPED FLOW

This boundary condition is applicable when the flow exits the domain into a long pipe or channel, at the end of which a flow profile is fully developed.

The channel can be thought of as a virtual extrusion of the outlet cross section. The outlet boundary must hence be flat in order for the fully developed flow condition to work properly. In 2D axisymmetric models, the outlet normal must be parallel to the symmetry axis.

Select an option to control the flow rate at the outlet:


- **Average velocity**, U_{av} .
- **Flow rate**, V_0 . Two-dimensional models also require an **Entrance thickness**, D_z , which is the out-of-plane thickness of the extruded entrance channel.
- **Average pressure**, P_{av} . Observe that P_{av} is the average pressure on the outflow.

The **Apply condition on each disjoint selection separately** check box is selected per default. When this setting is selected, the fully developed flow condition is applied separately on each disjoint selection. If this option is not selected, the condition is applied over the whole feature selection. The **Apply condition on each disjoint selection separately** should be disabled only if the flow conditions are known to be identical on each disjoint boundary.

The fully developed flow condition requires any volume force to be approximately aligned with the normal of the outlet boundary. The exception is gravity when the **Include gravity** setting is selected in the physics interface settings. Unless **Use reduced pressure** is also selected, an option to **Compensate for hydrostatic pressure** or **Compensate for hydrostatic pressure approximation** becomes available. It is selected per default and should only be deselected if the outlet normal is aligned with the gravity force and you want to specify an average pressure that includes the hydrostatic pressure.

A fully developed flow boundary cannot be adjacent to any Interior Wall feature.

CONSTRAINT SETTINGS

This section is displayed by clicking the **Show** button () and selecting **Advanced Physics Options**.



Prescribing Inlet and Outlet Conditions



Symmetry

This node provides a boundary condition for symmetry boundaries. It should only be used when the geometry and expected solution have mirror symmetry. By using symmetries in a model its size can be reduced by one-half or more, making this an efficient tool for solving large problems.

The **Symmetry** boundary condition prescribes no penetration and vanishing shear stresses. The boundary condition is a combination of a Dirichlet condition and a Neumann condition:

$$\mathbf{u} \cdot \mathbf{n} = 0, \quad \left(-p\mathbf{I} + \left(\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} \right) \right) \mathbf{n} = \mathbf{0}$$

$$\mathbf{u} \cdot \mathbf{n} = 0, \quad (-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T))\mathbf{n} = \mathbf{0}$$

for the compressible and incompressible formulations. The Dirichlet condition takes precedence over the Neumann condition, and the above equations are equivalent to the following equation for both the compressible and incompressible formulations:


$$\mathbf{u} \cdot \mathbf{n} = 0, \quad \mathbf{K} - (\mathbf{K} \cdot \mathbf{n})\mathbf{n} = \mathbf{0}$$

$$\mathbf{K} = \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)\mathbf{n}$$

BOUNDARY SELECTION

For 2D axial symmetry, a boundary condition does not need to be defined for the symmetry axis at $r = 0$. The software automatically provides a condition that prescribes $u_r = 0$ and vanishing stresses in the z direction and adds an **Axial Symmetry** node that implements these conditions on the axial symmetry boundaries only.

CONSTRAINT SETTINGS

This section is displayed by clicking the **Show** button () and selecting **Advanced Physics Options**.

Open Boundary

The **Open Boundary** condition describes boundaries in contact with a large volume of fluid. Fluid can both enter and leave the domain on boundaries with this type of condition.

BOUNDARY CONDITIONS

The **Boundary condition** options for open boundaries are **Normal stress** and **No viscous stress**.

Normal Stress

The **Normal stress** f_0 condition implicitly imposes $p \approx f_0$.

No Viscous Stress

The **No Viscous Stress** condition specifies vanishing viscous stress on the boundary. This condition does not provide sufficient information to fully specify the flow at the open boundary and must at least be combined with pressure constraints at adjacent points.

The **No viscous stress** condition prescribes:

$$\left(\mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} \right) \mathbf{n} = \mathbf{0}$$

$$\mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T) \mathbf{n} = \mathbf{0}$$

for the compressible and the incompressible formulations. This condition can be useful in some situations because it does not impose any constraint on the pressure. A typical example is a model with volume forces that give rise to pressure gradients that are hard to prescribe in advance. To make the model numerically stable, this boundary condition should be combined with a point constraint on the pressure.

Boundary Stress

The **Boundary Stress** node adds a boundary condition that represents a general class of conditions also known as traction boundary conditions.

BOUNDARY CONDITION

The **Boundary condition** options for the boundary stress are **General stress**, **Normal stress**, and **Normal stress, normal flow**.

General Stress

When **General stress** is selected, the components for the **Stress** \mathbf{F} should be specified. The total stress on the boundary is set equal to the given stress \mathbf{F} :

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

$$(-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T))\mathbf{n} = \mathbf{F}$$

for the compressible and the incompressible formulations.

This boundary condition implicitly sets a constraint on the pressure that for 2D flows is

$$p = 2\mu \frac{\partial u_n}{\partial n} - \mathbf{n} \cdot \mathbf{F} \quad (3-1)$$

If $\partial u_n / \partial n$ is small, [Equation 3-1](#) states that $p \approx -\mathbf{n} \cdot \mathbf{F}$.

Normal Stress

Normal Stress is described for the Open Boundary node.

Normal Stress, Normal Flow

For **Normal stress, normal flow**, the magnitude of the **Normal stress** f_0 should be specified. The tangential velocity is set to zero on the boundary:

$$\mathbf{n}^T \left(-p\mathbf{I} + \left(\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}\right)\right)\mathbf{n} = -f_0, \quad \mathbf{t} \cdot \mathbf{u} = 0$$

$$\mathbf{n}^T (-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T))\mathbf{n} = -f_0, \quad \mathbf{t} \cdot \mathbf{u} = 0$$


for the compressible and the incompressible formulations.

This boundary condition implicitly sets a constraint on the pressure that for 2D flows is

$$p = 2\mu \frac{\partial u_n}{\partial n} + f_0 \quad (3-2)$$

If $\partial u_n / \partial n$ is small, [Equation 3-2](#) states that $p \approx f_0$.

CONSTRAINT SETTINGS

This section is displayed by clicking the **Show** button () and selecting **Advanced Physics Options**.

If **Normal Stress**, **Normal Flow** is selected as the **Boundary condition**, then to **Apply reaction terms on all dependent variables**, the **All physics (symmetric)** option should be selected. Alternatively, the **Individual dependent variables** could be selected to restrict the reaction terms as needed.

Periodic Flow Condition

The **Periodic Flow Condition** splits its selection into a source group and a destination group. Fluid that leaves the domain through one of the destination boundaries enters the domain through the corresponding source boundary. This corresponds to a situation where the geometry is a periodic part of a larger geometry. If the boundaries are not parallel to each other, the velocity vector is automatically transformed.

If the boundaries are curved, the orientation of the source must be specified manually (see [Orientation of Source](#)).

No input is required when **Compressible flow (Ma<0.3)** is selected for **Compressibility** under the **Physical Model** section for the physics interface. Typically when a periodic boundary condition is used with a compressible flow, the pressure is the same at both boundaries and the flow is driven by a volume force.


FLOW CONDITION

This section is available when **Incompressible flow** is selected for **Compressibility** under the **Physical Model** section for the physics interface.

The **Flow Condition** at the boundary is specified through a value or expression for either the **Pressure difference**, $p_{\text{src}} - p_{\text{dst}}$, or the **Mass flow**, \dot{m} . The mass flow option ensures a pressure difference is achieved across the source and destination boundaries. This pressure difference can, for example, drive the fully developed flow in a channel.

To set up a periodic boundary condition, both boundaries must be selected in the **Periodic Flow Condition** node. COMSOL Multiphysics automatically assigns one boundary as the source and the other as the destination. To manually set the destination selection, a **Destination Selection** subnode is available from the context menu (by right-clicking the parent node) or from the **Physics** toolbar, **Attributes** menu. All destination sides must be connected.

CONSTRAINT SETTINGS

This section is displayed by clicking the **Show** button () and selecting **Advanced Physics Options**.

ORIENTATION OF SOURCE

For information about the **Orientation of Source** section, see [Orientation of Source and Destination](#) in the *COMSOL Multiphysics Reference Manual*.

Flow Continuity

The **Flow Continuity** condition is suitable for pairs where the boundaries match; it prescribes that the flow field is continuous across the pair.

The parts of the boundary where a source boundary lacks a corresponding destination boundary and vice versa will get conditions from other boundary conditions under the physics interface acting as fallback boundary conditions on the nonoverlapping parts. By default, the [Wall](#) node is used as fallback unless another feature has been selected for the boundary.



[About Identity and Contact Pairs](#)


Pressure Point Constraint

The **Pressure Point Constraint** condition can be used to specify the pressure level. If it is not possible to specify the pressure level using a boundary condition, the pressure level must be set in some other way, for example, by specifying a fixed pressure at a point.

PRESSURE CONSTRAINT

The relative pressure value is set by specifying the **Pressure** p_0 . If the reference pressure p_{ref} defined at the physics interface level is equal to zero, p_0 represents the absolute pressure.

CONSTRAINT SETTINGS

This section is displayed by clicking the **Show** button () and selecting **Advanced Physics Options**.

The **Apply reaction terms on** setting is set per default to **Individual dependent variables**. **All physics (symmetric)** cannot be used together with a segregated solver when fluid flow is coupled with Moving Mesh, Level Set, or Phase Field.

Point Mass Source

This feature requires at least one of the following licenses: Battery Design Module, CFD Module, Chemical Reaction Engineering Module, Corrosion Module, Electrochemistry Module, Electrodeposition Module, Fuel Cell & Electrolyzer Module, Microfluidics Module, Pipe Flow Module, or Subsurface Flow Module.

The **Point Mass Source** feature models mass flow originating from an infinitely small domain centered around a point.

For the Reacting Flow in Porous Media, Diluted Species interface, which is available with the CFD Module, Chemical Reaction Engineering Module, or Battery Design Module, there are two nodes: one for the fluid flow (**Fluid Point Source**) and one for the species (**Species Point Source**).

SOURCE STRENGTH

The source **Mass flux**, \dot{q}_p , should be specified. A positive value results in mass being ejected from the point into the computational domain. A negative value results in mass being removed from the computational domain.

Point sources located on a boundary or on an edge affect the adjacent computational domains. This has the effect, for example, that a point source located on a symmetry plane has twice the given strength.



Mass Sources for Fluid Flow in the *COMSOL Multiphysics Reference Manual*

Line Mass Source

This feature requires at least one of the following licenses: Battery Design Module, CFD Module, Chemical Reaction Engineering Module, Corrosion Module, Electrochemistry Module, Electrodeposition Module, Fuel Cell & Electrolyzer Module, Microfluidics Module, Pipe Flow Module, or Subsurface Flow Module.

The **Line Mass Source** feature models mass flow originating from a tube region with infinitely small radius.

For the Reacting Flow in Porous Media, Diluted Species interface, which is available with the CFD Module, Chemical Reaction Engineering Module, or Battery Design Module, there are two nodes, one for the fluid flow (**Fluid Line Source**) and one for the species (**Species Line Source**).

SELECTION

The **Line Mass Source** feature is available for all dimensions, but the applicable selection differs between the dimensions.

MODEL DIMENSION	APPLICABLE GEOMETRICAL ENTITY
2D	Points
2D Axisymmetry	Points not on the axis of symmetry
3D	Edges

SOURCE STRENGTH

The source **Mass flux**, \dot{q}_1 , should be specified. A positive value results in mass being ejected from the line into the computational domain and a negative value means that mass is removed from the computational domain.

Line sources located on a boundary affect the adjacent computational domains. This, for example, has the effect that a line source located on a symmetry plane has twice the given strength.



[Mass Sources for Fluid Flow](#) in the *COMSOL Multiphysics Reference Manual*

Gravity

The **Gravity** global feature is automatically added when **Include gravity** is selected at the interface level in the **Physical Model** settings. It defines the gravity forces from the **Acceleration of gravity** value. When a turbulence model that solves for the turbulent kinetic energy, k , is used, the option **Include buoyancy-induced turbulence** is available. When selected, the **Buoyancy contribution** is by default set to **Automatic from multiphysics**. Contributions are only obtained from multiphysics couplings that support buoyancy-induced turbulence, such as Nonisothermal Flow. If the **Buoyancy contribution** is switched to **User defined**, a text field for the **Turbulent Schmidt number** appears.

ACCELERATION OF GRAVITY

The **Acceleration of gravity** (SI unit m/s, default value $-g_{\text{const}}\mathbf{e}_z$ in 2D axial symmetry and 3D and $-g_{\text{const}}\mathbf{e}_y$ in 2D) is used to define the gravity forces. It should be a global quantity.

Theory for the Single-Phase Flow Interfaces

The theory for the Single-Phase Flow, Laminar Flow interface is described in this section:

- [General Single-Phase Flow Theory](#)
- [Compressible Flow](#)
- [Weakly Compressible Flow](#)
- [The Mach Number Limit](#)
- [Incompressible Flow](#)
- [The Reynolds Number](#)
- [Gravity](#)
- [Theory for the Wall Boundary Condition](#)
- [Prescribing Inlet and Outlet Conditions](#)
- [Fully Developed Flow \(Inlet\)](#)
- [Fully Developed Flow \(Outlet\)](#)
- [No Viscous Stress](#)
- [Normal Stress Boundary Condition](#)
- [Mass Sources for Fluid Flow](#)
- [Numerical Stability — Stabilization Techniques for Fluid Flow](#)
- [Solvers for Laminar Flow](#)
- [Pseudo Time Stepping for Laminar Flow Models](#)
- [Discontinuous Galerkin Formulation](#)
- [Particle Tracing in Fluid Flow](#)
- [References for the Single-Phase Flow, Laminar Flow Interfaces](#)



The theory about most boundary conditions is found in [Ref. 2](#).

The Single-Phase Fluid Flow interfaces are based on the Navier–Stokes equations, which in their most general form read

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (3-3)$$

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho (\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot [-p \mathbf{I} + \mathbf{K}] + \mathbf{F} \quad (3-4)$$

$$\rho C_p \left(\frac{\partial T}{\partial t} + (\mathbf{u} \cdot \nabla) T \right) = -(\nabla \cdot \mathbf{q}) + \mathbf{K} : \mathbf{S} - \frac{T}{\rho} \frac{\partial \rho}{\partial T} \bigg|_p \left(\frac{\partial p}{\partial t} + (\mathbf{u} \cdot \nabla) p \right) + Q \quad (3-5)$$

where

- ρ is the density (SI unit: kg/m³)
- \mathbf{u} is the velocity vector (SI unit: m/s)
- p is pressure (SI unit: Pa)
- \mathbf{I} is the identity matrix (unitless)
- \mathbf{K} is the viscous stress tensor (SI unit: Pa)
- \mathbf{F} is the volume force vector (SI unit: N/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{q} is the heat flux vector (SI unit: W/m²)
- Q contains the heat sources (SI unit: W/m³)
- \mathbf{S} is the strain-rate tensor:

$$\mathbf{S} = \frac{1}{2}(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)$$

The operation “:” denotes a contraction between tensors defined by

$$\mathbf{a} : \mathbf{b} = \sum_n \sum_m a_{nm} b_{nm} \quad (3-6)$$

This is sometimes referred to as the double dot product.

Equation 3-3 is the continuity equation and represents conservation of mass.

Equation 3-4 is a vector equation which represents conservation of momentum.

Equation 3-5 describes the conservation of energy, formulated in terms of temperature. This is an intuitive formulation that facilitates boundary condition specifications.

To close the equation system, Equation 3-3 through Equation 3-5, constitutive relations are needed.

For a Newtonian fluid, which has a linear relationship between stress and strain, Stokes (Ref. 1) deduced the following expression:

$$\mathbf{K} = 2\mu\mathbf{S} - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} \quad (3-7)$$

The dynamic viscosity, μ (SI unit: Pa·s), for a Newtonian fluid is allowed to depend on the thermodynamic state but not on the velocity field. All gases and many liquids can be considered Newtonian.

For an inelastic non-Newtonian fluid, the relationship between stress and strain rate is nonlinear, and an apparent viscosity is introduced instead of the dynamic viscosity. Examples of non-Newtonian fluids are honey, mud, blood, liquid metals, and most polymer solutions.

In theory, the same equations describe both laminar and turbulent flows. In practice, however, the mesh resolution required to simulate turbulence with the Laminar Flow interface makes such an approach impractical.



There are several books where derivations of the Navier–Stokes equations and detailed explanations of concepts such as Newtonian fluids can be found. See, for example, the classical text by Batchelor (Ref. 3) and the more recent work by Panton (Ref. 4).

Many applications describe isothermal flows for which Equation 3-5 is decoupled from Equation 3-3 and Equation 3-4.

2D AXISYMMETRIC FORMULATIONS

A 2D axisymmetric formulation of Equation 3-3 and Equation 3-4 requires $\partial/\partial\phi$ to be zero. That is, there must be no gradients in the azimuthal direction. A common additional assumption is, however, that $u_\phi = 0$. In such cases, the ϕ -equation can be removed from Equation 3-4. The resulting system of equations is both easier to converge and computationally less expensive compared to retaining the ϕ -equation.

The default 2D axisymmetric formulation of [Equation 3-3](#) and [Equation 3-4](#) therefore assumes that

$$\begin{aligned}\partial/\partial\phi &= 0 \\ u_\phi &= 0\end{aligned}$$

Compressible Flow

The equations of motion for a single-phase fluid are the continuity equation:

$$\frac{\partial\rho}{\partial t} + \nabla \cdot (\rho \mathbf{u}) = 0 \quad (3-8)$$

and the momentum equation:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p + \nabla \cdot \left(\mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right) + \mathbf{F} \quad (3-9)$$

These equations are applicable for incompressible as well as for compressible flow with density and viscosity variations.

Weakly Compressible Flow

The same equations as for [Compressible Flow](#) are applied for weakly compressible flow. The only difference is that the density is evaluated at the reference pressure. The density may be a function of other quantities, in particular it may be temperature dependent.

The weakly compressible flow equations are valid for incompressible as well as compressible flow with density variations independent of the pressure.

Provided that the densities dependency pressure is specified through model inputs, the density is automatically evaluated at the reference pressure level.

The Mach Number Limit

An important dimensionless number in fluid dynamics is the Mach number, Ma , defined by

$$\text{Ma} = \frac{|\mathbf{u}|}{a}$$

where a is the speed of sound. A flow is formally incompressible when $\text{Ma} = 0$. This is theoretically achieved by letting the speed of sound tend to infinity. The Navier–Stokes equations then have the mathematical property that pressure disturbances are instantaneously propagated throughout the entire domain. This results in a parabolic equation system.

The momentum equation, Equation 3-9, is parabolic for unsteady flow and elliptic for steady flow, whereas the continuity equation, Equation 3-8, is hyperbolic for both steady and unsteady flow. The combined system of equations is thus hybrid parabolic-hyperbolic for unsteady flow and hybrid elliptic-hyperbolic for steady flow. An exception occurs when the viscous term in Equation 3-9 becomes vanishingly small, such as at an outflow boundary, in which case the momentum equation becomes locally hyperbolic. The number of boundary conditions to apply on the boundary then depends on the number of characteristics propagating into the computational domain. For the purely hyperbolic system, the number of characteristics propagating from the boundary into the domain changes as the Mach number passes through unity. Hence, the number of boundary conditions required to obtain a numerically well-posed system must also change. The compressible formulation of the laminar and turbulent interfaces uses the same boundary conditions as the incompressible formulation, which implies that the compressible interfaces are not suitable for flows with a Mach number larger than or equal to one. Yet, the practical Mach number limit is lower than one. The main reason is that the numerical scheme (stabilization and boundary conditions) of the Laminar Flow interface does not recognize the direction and speed of pressure waves. The fully compressible Navier–Stokes equations do, for example, start to display very sharp gradients already at moderate Mach numbers. But the stabilization for the single-phase flow interface does not necessarily capture these gradients. It is impossible to give an exact limit where the low Mach number regime ends and the moderate Mach number regime begins, but a rule of thumb is that the Mach number effects start to appear at $\text{Ma} = 0.3$. For this reason, the compressible formulation is referred to as *Compressible flow* ($\text{Ma} < 0.3$) in COMSOL Multiphysics.

Incompressible Flow

When the temperature variations in the flow are small, a single-phase fluid can often be assumed incompressible; that is, ρ is constant or nearly constant. This is the case for all liquids under normal conditions and also for gases at low velocities. For constant ρ , Equation 3-8 reduces to

$$\rho \nabla \cdot \mathbf{u} = 0 \quad (3-10)$$

and Equation 3-9 becomes

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho(\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot [-p \mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \mathbf{F} \quad (3-11)$$

Provided that the densities dependency on temperature and pressure is specified through model inputs, the density is evaluated at the reference pressure level and at the reference temperature. However, if the density is a function of other quantities such as a concentration field, or if the density is specified by a user defined expression, the user has to make sure that the density is defined as constant when the incompressible flow formulation is used.

The Reynolds Number

A fundamental characteristic in analyses of fluid flow is the Reynolds number:

$$\text{Re} = \frac{\rho U L}{\mu}$$

where U denotes a velocity scale, and L denotes a representative length. The Reynolds number represents the ratio between inertial and viscous forces. At low Reynolds numbers, viscous forces dominate and tend to damp out all disturbances, which leads to laminar flow. At high Reynolds numbers, the damping in the system is very low, giving small disturbances the possibility to grow by nonlinear interactions. If the Reynolds number is high enough, the flow field eventually ends up in a chaotic state called turbulence.

Note that the Reynolds number can have different meanings depending on the length scale and velocity scale. To be able to compare two Reynolds numbers, they must be based on equivalent length and velocity scales.

The Fluid Flow interfaces automatically calculate the local cell Reynolds number $\text{Re}^c = \rho |\mathbf{u}| h / (2\mu)$ using the element length h for L and the magnitude of the velocity vector \mathbf{u} for the velocity scale U . This Reynolds number is not related to the character of the flow field, but to the stability of the numerical discretization. The risk for numerical oscillations in the solution increases as Re^c grows. The cell Reynolds number is a predefined quantity available for visualization and evaluation (typically it is available as: `spf.cellRe`).

DEFINITION

The gravity force is defined from the acceleration of gravity vector, \mathbf{g} , and the density, ρ . Under usual conditions and in Cartesian coordinates with the z -axis in the vertical direction,

$$\mathbf{g} = \begin{pmatrix} 0 \\ 0 \\ -g_{\text{const}} \end{pmatrix}.$$

When gravity is considered, a volume force equal to $\rho\mathbf{g}$ is included in the momentum equation. For example, for laminar weakly compressible flow, it reads:

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = \nabla \cdot \left(-p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right) + \mathbf{F} + \rho \mathbf{g} \quad (3-12)$$

Introducing a constant reference density ρ_{ref} , and assuming that \mathbf{g} is homogeneous, this equation is equivalently written:

$$\begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = & \\ \nabla \cdot \left(-p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} + \rho_{\text{ref}} \mathbf{g} \cdot (\mathbf{r} - \mathbf{r}_{\text{ref}}) \mathbf{I} \right) + \mathbf{F} + (\rho - \rho_{\text{ref}}) \mathbf{g} & \end{aligned} \quad (3-13)$$

where \mathbf{r} is the position vector and \mathbf{r}_{ref} is an arbitrary reference position vector.

From this equation, it is convenient to define the reduced pressure which accounts for the hydrostatic pressure, $\tilde{p} = p - \rho_{\text{ref}} \mathbf{g} \cdot (\mathbf{r} - \mathbf{r}_{\text{ref}})$

$$\begin{aligned} \rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = \nabla \cdot \left(-\tilde{p} \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right) + & \\ \mathbf{F} + (\rho - \rho_{\text{ref}}) \mathbf{g} & \end{aligned} \quad (3-14)$$

In Equation 3-14, the gravity force is written $(\rho - \rho_{\text{ref}}) \mathbf{g}$.

PRESSURE FORMULATION

When the relative pressure is used (default option) the interface dependent variable represents the relative pressure and the absolute pressure is defined as $p_A = p_{\text{ref}} + p$. When the pressure is used to define a boundary condition (for example when p_0

defines the pressure condition at an outlet), it represents the relative pressure. Hence defining the outlet pressure as $p_{\text{hydro, approx}} = -\rho_{\text{ref}} \mathbf{g} \cdot (\mathbf{r} - \mathbf{r}_{\text{ref}})$ compensates for the gravity force for an ambient reference pressure of 0 Pa when the density is constant, there is no external force, and provided p_{ref} , \mathbf{g} and \mathbf{r}_0 are defined consistently.

When the reduced pressure is used, the interface dependent variable (named p by default) represents the reduced pressure. The absolute pressure is then defined as $p_A = p_{\text{ref}} - \rho_{\text{ref}} \mathbf{g} \cdot (\mathbf{r} - \mathbf{r}_{\text{ref}}) + p$. In this case when the pressure is used to define a boundary condition (for example to define a pressure condition at an outlet), its value corresponds to the reduced pressure. Hence, the prescribed pressure compensates for an approximate hydrostatic pressure, $p_{\text{hydro, approx}} = -\rho_{\text{ref}} \mathbf{g} \cdot (\mathbf{r} - \mathbf{r}_{\text{ref}})$ which is exact only when the density is constant and there is no external force.

PRESSURE BOUNDARY CONDITION

For an immobile fluid the momentum equation simplifies to $\nabla \cdot (p\mathbf{I}) = \mathbf{F} + \rho \mathbf{g}$ or

$$\nabla \cdot (-\tilde{p}\mathbf{I}) = \mathbf{F} + (\rho - \rho_{\text{ref}}) \mathbf{g} \text{ depending on the pressure formulation.}$$

For incompressible flow, assuming there are no external forces, this leads respectively to $p = -\rho_{\text{ref}} \mathbf{g} \cdot (\mathbf{r} - \mathbf{r}_{\text{ref}}) + p_0$ or $p = p_0$.

Theory for the Wall Boundary Condition

See [Wall](#) for the node settings.

SLIP

The [Slip](#) condition assumes that there are no viscous effects at the slip wall and hence, no boundary layer develops. From a modeling point of view, this is a reasonable approximation if the important effect of the wall is to prevent fluid from leaving the domain. Mathematically, the constraint can be formulated as:

$$\mathbf{u} \cdot \mathbf{n} = 0, \quad (-p\mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T))\mathbf{n} = \mathbf{0}$$

The no-penetration term takes precedence over the Neumann part of the condition and the above expression is therefore equivalent to

$$\begin{aligned} \mathbf{u} \cdot \mathbf{n} &= 0, & \mathbf{K}_n - (\mathbf{K}_n \cdot \mathbf{n})\mathbf{n} &= \mathbf{0} \\ \mathbf{K}_n &= \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)\mathbf{n} \end{aligned}$$

expressing that there is no flow across the boundary and no viscous stress in the tangential direction.

For a moving wall with translational velocity \mathbf{u}_{tr} , \mathbf{u} in the above equations is replaced by the relative velocity $\mathbf{u}_{rel} = \mathbf{u} - \mathbf{u}_{tr}$.

POROUS SLIP

The **Porous slip** option, which can be chosen in the **Porous treatment of no slip condition** list when **Enable porous media domains** is activated, results in special treatment of **Wall** boundaries and **Interior Wall** boundaries adjacent to porous domains (**Porous Medium** feature). Namely, similar to the **Navier slip** boundary condition, no penetration and tangential stress conditions are applied at the wall:

$$\mathbf{u} \cdot \mathbf{n} = 0, \quad \mathbf{K}_n - (\mathbf{K}_n \cdot \mathbf{n})\mathbf{n} = -\frac{\mu}{\varepsilon_p} \frac{\mathbf{u}_{slip}}{y_{ps}},$$

$$\mathbf{u}_{slip} = \mathbf{u} - (\mathbf{u} \cdot \mathbf{n})\mathbf{n}, \quad \mathbf{K}_n = \frac{\mu}{\varepsilon_p} (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)\mathbf{n}$$

Here, \mathbf{K}_n is the viscous wall traction, \mathbf{n} is the wall normal, \mathbf{u}_{slip} is the tangential velocity at the wall while the real no slip is assumed to be at a distance d_w (half-height of the first cell adjacent to the wall) outside the wall, and y_{ps} is the porous slip length. An analytical derivation of the velocity profile in the boundary layer where the pressure gradient is balanced by the sum of the Darcy term, Forchheimer drag, and a viscous term (neglecting convective terms) leads to the following expression for y_{ps} :

$$y_{ps} = \tilde{l} \left(\frac{1 + X_p e^{-\xi_p}}{1 - X_p e^{-\xi_p}} \right) \left[\left(\frac{1 + X_p e^{-\xi_p}}{1 + X_p} \right)^2 e^{\xi_p} - 1 \right], \quad \xi_p = \frac{d_w}{\tilde{l}}, \quad \tilde{l} = \frac{l_{pore}}{\sqrt{1 + 2c_\beta}},$$

$$X_p = \frac{\sqrt{1 + 2c_\beta} - \sqrt{1 + \frac{4}{3}c_\beta}}{\sqrt{1 + 2c_\beta} + \sqrt{1 + \frac{4}{3}c_\beta}}, \quad c_\beta = \frac{\sqrt{1 + 4c_D} - 1}{2}, \quad c_D = \frac{\beta_{ND}}{\mu} \frac{\kappa^2}{\mu} |\nabla p - \rho \mathbf{g}|$$

where $l_{pore} = \sqrt{\kappa/\varepsilon_p}$ is the porous length scale, $\beta_{ND} = \rho c_F/\sqrt{\kappa}$ is the non-Darcian coefficient (c_F is the Forchheimer coefficient), \mathbf{g} is the gravity vector, and $\xi_p, \tilde{l}, X_p, c_\beta, c_D$ are intermediate variables. This formula is used when the **Pressure-gradient formulation** is chosen and it reads pressure-gradient at the wall. By default, the **Velocity formulation** is activated and the corresponding formula is:

$$y_{ps} = \tilde{l} (e^{\xi_p} - 1), \quad \xi_p = \frac{d_w}{\tilde{l}}, \quad \tilde{l} = \frac{l_{\text{pore}}}{\sqrt{1 + 2c_\beta}},$$

$$c_\beta = \left[\left(\frac{\beta_{\text{ND}} \kappa |\mathbf{u}_{\text{slip}}|}{\mu} \frac{l_{\text{pore}}}{d_w} \right)^2 / 2 \right]^{1/3}$$

Although this formulation is an approximation, since it uses slip velocity at the wall to reconstruct the pressure gradient, it is rather accurate. Moreover, the influence of convective terms is partially accounted for in this formulation.

SLIDING WALL

The sliding wall option is appropriate if the wall behaves like a conveyor belt; that is, the surface is sliding in its tangential direction. The wall does not have to actually move in the coordinate system.

- In 2D, the tangential direction is unambiguously defined by the direction of the boundary, but the situation becomes more complicated in 3D. For this reason, this boundary condition has slightly different definitions in the different space dimensions.
- For 2D and 2D axisymmetric components, the velocity is given as a scalar U_w and the condition prescribes

$$\mathbf{u} \cdot \mathbf{n} = 0, \quad \mathbf{u} \cdot \mathbf{t} = U_w$$

where $\mathbf{t} = (n_y, -n_x)$ for 2D and $\mathbf{t} = (n_z, -n_r)$ for axial symmetry.

- For 3D components, the velocity is set equal to a given vector \mathbf{u}_w projected onto the boundary plane:

$$\mathbf{u} = \frac{\mathbf{u}_w - (\mathbf{n} \cdot \mathbf{u}_w) \mathbf{n}}{\|\mathbf{u}_w - (\mathbf{n} \cdot \mathbf{u}_w) \mathbf{n}\|} \|\mathbf{u}_w\|$$

The normalization makes \mathbf{u} have the same magnitude as \mathbf{u}_w even if \mathbf{u}_w is not exactly parallel to the wall.

CONSTRAINT SETTINGS

The wall feature uses three different techniques to constraint the velocity field:

- Pointwise constraints is the standard technique to enforce strong constraints in the finite element method. The desired value of the velocity is prescribed at each node point in the mesh. Since the constraint is enforced locally at each node, only local values are affected by the constraint and the constraints are independent of each other. The solvers can therefore eliminate both the constrained degrees of freedom

and the constraint force terms, effectively reducing the number of degrees of freedom being solved for.

The main advantage of pointwise constraints is that they enforce the constraint exactly pointwise. This means that they do not introduce any leakage of the velocity across the wall, unless specified. The main disadvantage of pointwise constraints is that they introduce locking effects when trying to impose a no-penetration condition for the velocity, $\mathbf{u} \cdot \mathbf{n} = 0$, on curved walls or walls with sharp corners.

- Using Weak constraints is an alternative method to prescribe the velocity. It consists on enforcing the boundary condition for the velocity via *Lagrange multipliers*. Their main advantage is that the Lagrange multiplier can provide an accurate representation of the reaction flux at the wall. Their main disadvantage is that they introduce extra unknowns, and are usually difficult to combine with other constraint methods on adjacent boundaries. Moreover, they may require extra constraints for the Lagrange multipliers. For more information, see [Weak Constraints](#) in the *COMSOL Multiphysics Reference Manual*.
- Discontinuous Galerkin (DG) constraints use a numerical flux to prescribe the velocity at the wall. They impose the constraint in an integral sense rather than pointwise, and do not suffer from the locking effects introduced by pointwise constraints when trying to prescribe a no penetration condition for the velocity. They are also better behaved when prescribing nonlinear constraints. Their main disadvantage is that the constraint is only imposed approximately, and may produce small leaks. For more information, see [Discontinuous Galerkin Formulation](#).

The following combination of Constraint techniques can be selected in the Constraint Setting sections of Wall boundary conditions:

- Use default settings. The default settings use different constraint methods depending on whether only the normal component of the velocity is prescribed, such as in the no penetration condition, $\mathbf{u} \cdot \mathbf{n} = 0$, imposed for example in Slip walls or No Slip walls using Wall Functions or Automatic Wall Treatment, or both tangential and normal components are prescribed, as is the case of No Slip walls in laminar flow.

DG constraints are used to impose the no penetration condition for Slip walls. When a No Slip condition is prescribed, pointwise constraints are used except for moving walls where DG constraints are used.

- Use Pointwise constraints.
- Use DG constraints.

- Use Weak constraints. Weak constraints are not available on Interior Walls.
- Use Mixed constraints. This option is only available when both the tangential and normal components of the velocity need to be prescribed. The velocity on the wall normal direction is imposed via pointwise constraints. The constraint for the tangential directions is relaxed, and DG constraints are used instead. This provides improved accuracy and performance when working with coarse boundary layer meshes. For more information, see [Ref. 18](#).

Prescribing Inlet and Outlet Conditions

The Navier–Stokes equations can show large variations in mathematical behavior, ranging from almost completely elliptic to almost completely hyperbolic. This has implications when it comes to prescribing admissible boundary conditions. There is also a discrepancy between mathematically valid boundary conditions and practically useful boundary conditions. See [Inlet](#) and [Outlet](#) for the node settings.

INLET CONDITIONS

An inlet requires specification of the velocity components. The most robust way to do this is to prescribe a velocity field using a Velocity condition.

A common alternative to prescribing the complete velocity field is to prescribe a pressure and all but one velocity component. The pressure cannot be specified pointwise because this is mathematically over-constraining. Instead the pressure can be specified via a stress condition:

$$-p + 2\mu \frac{\partial u_n}{\partial n} = F_n \quad (3-15)$$

where $\partial u_n / \partial n$ is the normal derivative of the normal velocity component.

[Equation 3-15](#) is prescribed by the Pressure condition in the Inlet and Outlet features and the Normal stress condition in the Open Boundary and Boundary Stress features. [Equation 3-15](#) is mathematically more stringent compared to specifying the pressure pointwise and at the same time cannot guarantee that p obtains the desired value. In practice, p is close to F_n , except for low Reynolds number flows where viscous effects are the only effects that balance the pressure. In addition to [Equation 3-15](#), all but one velocity component must be specified. For low Reynolds numbers, this can be specified by a vanishing tangential stress condition:

$$\mu \frac{\partial u_t}{\partial n} = 0$$

which is what the Normal stress condition does. Vanishing tangential stress becomes a less well-posed inlet condition as the Reynolds number increases. The Pressure condition in the Inlet feature therefore requires a flow direction to be prescribed, which provides a well-posed condition independent of Reynolds number.

OUTLET CONDITIONS

The most common approach is to prescribe a pressure via a normal stress condition on the outlet. This is often accompanied by a vanishing tangential stress condition:

$$\mu \frac{\partial u_t}{\partial n} = 0$$

where $\partial u_t / \partial n$ is the normal derivative of the tangential velocity field. It is also possible to prescribe u_t to be zero. The latter option should be used with care since it can have a significant effect on the upstream solution.

The elliptic character of the Navier-Stokes equations mathematically permit specifying a complete velocity field at an outlet. This can, however, be difficult to apply in practice. The reason being that it is hard to prescribe the outlet velocity so that it is consistent with the interior solution at each point. The adjustment to the specified velocity then occurs across an outlet boundary layer. The thickness of this boundary layer depends on the Reynolds number; the higher the Reynolds number, the thinner the boundary layer.

ALTERNATIVE FORMULATIONS

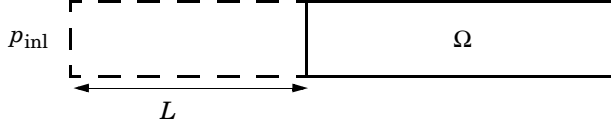
COMSOL provides several specialized boundary conditions that either provide detailed control over the flow at the boundary or that simulate specific devices. In practice they often prescribe a velocity or a pressure, but calculate the prescribed values using for example ODEs.



Normal Stress Boundary Condition

Fully Developed Flow (Inlet)

The **Fully developed flow** boundary condition can be understood from the following figure:



The flow to the domain Ω is assumed to enter through a straight channel of length L . The channel is a virtual extrusion of the inlet cross section and a pressure constant pressure P_{inl} is applied on the inlet of the virtual channel.

The Fully Developed Flow boundary condition prescribes that the tangential flow component on the boundary is zero:

$$\mathbf{u} - (\mathbf{u} \cdot \mathbf{n})\mathbf{n} = \mathbf{0}$$

The momentum equation for a fully developed flow in the virtual extrusion of the inlet cross section can then be projected onto the inlet boundary with the following weak equation as the result:

$$(-\mu(\nabla_{\mathbf{t}}\mathbf{u} + (\nabla_{\mathbf{t}}\mathbf{u})^T) + (p + P_{\text{inl}})/2)\nabla_{\mathbf{t}}\hat{\mathbf{u}}L - P_{\text{inl}}(\mathbf{n} \cdot \hat{\mathbf{u}})$$

The exact value of L is somewhat arbitrary as long as it is not too high or too low. L is therefore set to ten times the inlet edge length in 2D and to ten times the square root of the inlet area in 2D axisymmetry and in 3D.

The fact that the velocity profile is not prescribed, but rather the solution of a projected weak contribution, means that the actual velocity profile that is obtained on the inlet can deviate from the analytical fully developed flow profile (in cases such an analytical solution exists) if required by the solution inside the computational domain, Ω . This is most notably if the flow is strongly curved just downstream of the inlet. This can for example be the case if an object is positioned just downstream of the inlet or if the adjacent boundaries are not orthogonal to the inlet. The analytical solution can in these cases be recovered by including a little bit of the inlet channel in the computational domain.

The inlet pressure, P_{inl} , must be solved for and its equation is a discrete algebraic equation (DAE) for P_{inl} . When, for example, the average velocity is specified, the DAE reads

$$(\langle \mathbf{u} \cdot \mathbf{n} \rangle + U_{av}) \hat{P}_{inl}$$

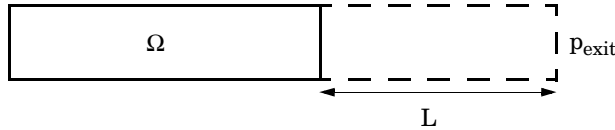
where $\langle \cdot \rangle$ denotes the average over the inlet. Since the equation for P_{inl} is a DAE (the equation for P_{inl} does not contain P_{inl}), it must be solved coupled to Navier–Stokes and it must be treated by a Vanka pre-smoother and post-smoother if iterative solvers are used.

The boundary conditions for the virtual inlet channel are inherited from the boundaries adjacent to the inlet channel. Virtual boundaries adjacent to walls (except slip walls) are treated as no-slip walls. Virtual boundaries adjacent to any other type of boundary are treated as slip-walls (or equivalently as symmetry boundaries). So in the figure above, if the lower boundary of Ω is a no-slip wall and the top boundary is a symmetry boundary, the lower boundary of the virtually extruded channel becomes a no-slip wall and the upper boundary becomes a symmetry boundary.

Algebraic turbulence models need additional equations or constraints. For all other turbulence models, their weak equations in the virtual inlet channel are projected onto the inlet of the computational domain. The projections are regularized to prevent the production to fall to zero, so the solutions of the projects can show slight deviation from the correct solutions close to no-slip walls.

Fully Developed Flow (Outlet)

The **Fully developed flow** boundary condition can be understood from the following figure:



The flow to the domain Ω is assumed to enter through a straight channel of length L . The channel is a virtual extrusion of the inlet cross section and a pressure constant pressure P_{exit} is applied on the inlet of the virtual channel.

The Fully Developed Flow boundary condition prescribes that the tangential flow component on the boundary is zero:

$$\mathbf{u} - (\mathbf{u} \cdot \mathbf{n})\mathbf{n} = \mathbf{0}$$

The momentum equation for a fully developed flow in the virtual extrusion of the inlet cross section can then be projected onto the inlet boundary with the following weak equation as the result:

$$(-\mu(\nabla_{\mathbf{t}}\mathbf{u} + (\nabla_{\mathbf{t}}\mathbf{u})^T) + (p + P_{\text{exit}})/2)\nabla_{\mathbf{t}}\hat{\mathbf{u}}L - P_{\text{exit}}(\mathbf{n} \cdot \hat{\mathbf{u}})$$

The exact value of L is somewhat arbitrary as long as it is not too high or too low. L is therefore set to ten times the inlet edge length in 2D and to ten times the square root of the inlet area in 2D axisymmetry and in 3D.

The fact that the velocity profile is not prescribed, but rather the solution of a projected weak contribution, means that the actual velocity profile that is obtained on the outlet can deviate from the analytical fully developed flow profile (in cases such an analytical solution exists) if required by the solution inside the computational domain, Ω . The inlet pressure, P_{exit} , must be solved for and the its equation is a discrete algebraic equation (DAE) for P_{exit} . When, for example, the average velocity is specified, the DAE reads

$$(\langle \mathbf{u} \cdot \mathbf{n} \rangle + U_{\text{av}})\hat{P}_{\text{exit}}$$

where $\langle \cdot \rangle$ denotes the average over the inlet. Since the equation for P_{ext} is a DAE, it must be solved coupled to Navier–Stokes and it must be treated by a Vanka pre-smoother and post-smoother if iterative solvers are used.

The boundary conditions for the virtual outlet channel are inherited from the boundaries adjacent to the outlet channel. Virtual boundaries adjacent to walls (except slip walls) are treated as no-slip walls. Virtual boundaries adjacent to any other type of boundary are treated as slip-walls (or equivalently as symmetry boundaries).

No Viscous Stress

For this module, and in addition to the Pressure, No Viscous Stress boundary condition, the viscous stress condition sets the viscous stress to zero:

$$\left(\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}\right)\mathbf{n} = \mathbf{0}$$

$$(\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T))\mathbf{n} = \mathbf{0}$$

using the compressible/weakly compressible and the incompressible formulation, respectively.

The condition is not a sufficient outlet condition since it lacks information about the outlet pressure. It must hence be combined with pressure point constraints on one or several points or lines surrounding the outlet.

This boundary condition is numerically the least stable outlet condition, but can still be beneficial if the outlet pressure is nonconstant due to, for example, a nonlinear volume force.

Normal Stress Boundary Condition

The total stress on the boundary is set equal to a stress vector of magnitude f_0 , oriented in the negative normal direction:

$$\left(-p\mathbf{I} + \left(\mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}\right)\right)\mathbf{n} = -f_0\mathbf{n}$$

$$(-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T))\mathbf{n} = -f_0\mathbf{n}$$

using the compressible/weakly compressible and the incompressible formulation, respectively.

This implies that the total stress in the tangential direction is zero. This boundary condition implicitly sets a constraint on the pressure which for 2D flows is

$$p = 2\mu \frac{\partial u_n}{\partial n} + f_0 \quad (3-16)$$

If $\partial u_n / \partial n$ is small, [Equation 3-16](#) states that $p \approx f_0$.

The Normal Stress condition is the mathematically correct version of the [Pressure Conditions](#) condition ([Ref. 4](#)), but it is numerically less stable.

Pressure Boundary Condition

For single-phase flow, a mathematically correct natural boundary condition for outlets is

$$\left(-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I}\right)\mathbf{n} = -p_0\mathbf{n} \quad (3-17)$$

$$(-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T))\mathbf{n} = -p_0\mathbf{n} \quad (3-18)$$

using the compressible/weakly compressible and the incompressible formulation, respectively.

This is a normal stress condition together with a no-tangential-stress condition. When $\mu > 0$, Equation 3-17 or Equation 3-18 can be supplemented with a tangential velocity condition

$$\mathbf{u} \cdot \mathbf{t} = 0 \quad (3-19)$$

If so, the no-tangential-stress condition is overridden. An issue with Equation 3-17 or Equation 3-18 is that it does not strongly enforce unidirectional flow on the boundary. If the prescribed pressure on an outlet is too high, parts of the outlet can actually have inflow. This is not as much of an issue for the Navier–Stokes equations as it is an issue for scalar transport equations solved along with the Navier–Stokes equations. Hence, when applying the **Pressure** boundary condition at an outlet or inlet you can further constrain the flow. With the **Suppress backflow** option

$$\begin{aligned} \left(-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} \right) \mathbf{n} &= -\hat{p}_0 \mathbf{n} \\ (-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)) \mathbf{n} &= -\hat{p}_0 \mathbf{n} \\ \hat{p}_0 &\leq p_0 \end{aligned} \quad , \quad (3-20)$$

the normal stress is adjusted to keep

$$\mathbf{u} \cdot \mathbf{n} \geq 0 \quad (3-21)$$

Equation 3-20 effectively means that the prescribed pressure is p_0 if $\mathbf{u} \cdot \mathbf{n} \geq 0$, but smaller at locations where $\mathbf{u} \cdot \mathbf{n} < 0$. This means that Equation 3-20 does not completely prevent backflow, but the backflow is substantially reduced. Backflow is suppressed also when external forces are acting on the fluid, provided the magnitude of these forces are of the same order as the dynamic pressure at the outlet.

A pressure condition can also be applied at an inlet. In this case, either the normal stress is prescribed

$$\begin{aligned} \mathbf{n}^T \left(-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T) - \frac{2}{3}\mu(\nabla \cdot \mathbf{u})\mathbf{I} \right) &= -\hat{p}_0 \\ \mathbf{n}^T (-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)) &= -\hat{p}_0 \\ \hat{p}_0 &\geq p_0 \end{aligned} \quad (3-22)$$

together with the tangential condition in [Equation 3-19](#), or, a general flow direction is prescribed.

$$\begin{aligned}
 \mathbf{r}_{\mathbf{u}}^T \left(-p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right) \mathbf{n} &= -\hat{p}_0 (\mathbf{r}_{\mathbf{u}} \cdot \mathbf{n}) \\
 \mathbf{r}_{\mathbf{u}}^T (-p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) \mathbf{n} &= -\hat{p}_0 (\mathbf{r}_{\mathbf{u}} \cdot \mathbf{n}) \\
 \hat{p}_0 &\geq p_0 \\
 \mathbf{u} - (\mathbf{u} \cdot \mathbf{r}_{\mathbf{u}}) \mathbf{r}_{\mathbf{u}} &= \mathbf{0}, \quad \mathbf{r}_{\mathbf{u}} = \frac{\mathbf{d}_{\mathbf{u}}}{\|\mathbf{d}_{\mathbf{u}}\|}
 \end{aligned} \tag{3-23}$$

The “>” option is used with suppress backflow to have $\mathbf{u} \cdot \mathbf{n} \leq 0$ or $\mathbf{u} \cdot \mathbf{r}_{\mathbf{u}} \geq 0$.

For incompressible single-phase flow, it is also allowed to specify the total pressure, p_{tot} , instead of the static pressure, p_{stat} , on inlet and outlet boundaries. It is more useful, for example, in pump applications. The pressure is then prescribed at the boundaries using Bernoulli’s principle,

$$p = p_{\text{stat}} = p_{\text{tot}} - \frac{1}{2} \rho |\mathbf{u}|^2 \tag{3-24}$$

The equation is imposed with two options: **Average** and **Pointwise**.

In the first option, p_{stat} is prescribed by:

$$\mathbf{n}^T (-p \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) \mathbf{n} = \mathbf{n}^T (-p_{\text{stat}} \mathbf{I} + \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T)) \mathbf{n} \tag{3-25}$$

where p_{tot} and $|\mathbf{u}|^2$ are averaged over the boundaries using the aveop operator:

$$p_{\text{stat}} = \text{aveop}(p_{\text{tot}}) - \frac{1}{2} \rho \cdot \text{aveop}(|\mathbf{u}|^2)$$

For the second option, [Equation 3-24](#) is prescribed pointwise.

See [Inlet](#), [Outlet](#), [Open Boundary](#), and [No Viscous Stress](#) for the individual node settings. Note that some modules have additional theory sections describing options available with that module.

Mass Sources for Fluid Flow

There are two types of mass sources in a Single-Phase Flow interface: point sources and line sources.



These features require at least one of the following licenses: Battery Design Module, CFD Module, Chemical Reaction Engineering Module, Corrosion Module, Electrochemistry Module, Electrodeposition Module, Fuel Cell & Electrolyzer Module, Microfluidics Module, Pipe Flow Module, Polymer Flow Module, or Subsurface Flow Module.

POINT SOURCE

A point source is theoretically formed by taking a mass injection/ejection, \dot{Q} (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$), in a small volume δV and then letting the size of the volume tend to zero while keeping the total mass flux constant. Given a point source strength, \dot{q}_p (SI unit: kg/s), this can be expressed as

$$\lim_{\delta V \rightarrow 0} \int_{\delta V} \dot{Q} = \dot{q}_p \quad (3-26)$$

An alternative way to form a point source/sink is to assume that mass is injected/extracted through the surface of a small object. Letting the object surface area tend to zero while keeping the mass flux constant, results in the same point source. For this alternative approach, effects resulting from the physical object volume, such as drag and fluid displacement, need to be neglected.

The weak contribution

$$\dot{q}_{p\text{test}}(p)$$

is added to a point in the geometry. As can be seen from [Equation 3-26](#), \dot{Q} must tend to plus or minus infinity as δV tends to zero. This means that in theory the pressure also tends to plus or minus infinity.

Observe that “point” refers to the physical representation of the source. A point source can therefore only be added to points in 3D components and to points on the symmetry axis in 2D axisymmetry components. Other geometrical points in 2D components represent physical lines.

The finite element representation of [Equation 3-26](#) corresponds to a finite pressure in a point with the effect of the point source spread out over a region around the point.

The size of the region depends on the mesh and on the strength of the source. A finer mesh gives a smaller affected region, but also a more extreme pressure value. It is important not to mesh too finely around a point source since the resulting pressure can result in unphysical values for the density, for example. It can also have a negative effect on the condition number for the equation system.

LINE SOURCE

A line source can theoretically be formed by assuming a source of strength \dot{Q} (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$), located within a tube with cross-sectional area δS and then letting δS tend to zero, while keeping the total mass flux per unit length constant. Given a line source strength, \dot{q}_l (SI unit: $\text{kg}/(\text{m} \cdot \text{s})$), this can be expressed as

$$\lim_{\delta S \rightarrow 0} \int_{\delta S} \dot{Q} = \dot{q}_l \quad (3-27)$$

As in the point source case, an alternative approach is to assume that mass is injected/extracted through the surface of a small object. This results in the same mass source, but requires that effects on the fluid resulting from the physical object volume are neglected.

The weak contribution

$$\dot{q}_{l\text{test}}(p)$$

is added to lines in 3D or to points in 2D (which represent cut-through views of lines). Line sources can also be added to the axisymmetry line in 2D axisymmetry components. It cannot, however, be added to geometrical lines in 2D because they represent physical planes.

As with a point source, it is important not to mesh too finely around the line source.




For feature node information, see [Line Mass Source](#) and [Point Mass Source](#) in the *COMSOL Multiphysics Reference Manual*.



For the Reacting Flow in Porous Media, Diluted Species interface, which is available with the CFD Module, Chemical Reaction Engineering Module, or Battery Design Module, these shared physics nodes are renamed as follows:

- The **Line Mass Source** node is available as two nodes, one for the fluid flow (**Fluid Line Source**) and one for the species (**Species Line Source**).
- The **Point Mass Source** node is available as two nodes, one for the fluid flow (**Fluid Point Source**) and one for the species (**Species Point Source**).

Numerical Stability — Stabilization Techniques for Fluid Flow

The momentum equation ([Equation 3-9](#) or [Equation 3-11](#)) is a (nonlinear) convection-diffusion equation. Such equations can easily become unstable if discretized using the Galerkin finite element method. Stabilized finite element methods are usually necessary in order to obtain physical solutions. The stabilization settings are found in the main Fluid Flow interface features. To display this section, click the **Show More Options** button () and select **Stabilization** in the **Show More Options** dialog box.

There are three types of stabilization methods available for Navier–Stokes — *streamline diffusion*, *crosswind diffusion*, and *isotropic diffusion*. Streamline diffusion and crosswind diffusion are consistent stabilization methods, whereas isotropic diffusion is an inconsistent stabilization method.

For optimal functionality, the exact weak formulations of and constants in the streamline diffusion and crosswind diffusion methods depend on the order of the shape functions (basis functions) for the elements. The values of constants in the streamline diffusion and crosswind diffusion methods follow [Ref. 5](#) and [Ref. 6](#).

STREAMLINE DIFFUSION

For strongly coupled systems of equations, the streamline diffusion method must be applied to the system as a whole rather than to each equation separately. These ideas were first explored by Hughes and Mallet ([Ref. 8](#)) and were later extended to Galerkin least-squares (GLS) applied to the Navier–Stokes equations ([Ref. 9](#)). This is the streamline diffusion formulation that COMSOL Multiphysics supports. The time-scale tensor is the diagonal tensor presented in [Ref. 10](#).

The time-scale tensors for time-dependent problems should in theory depend on a time-scale that many references just set to the time-step taken by the time-solver, Δt

(see for example [Ref. 9](#) and [Ref. 10](#)). This simple approach does not, however, necessarily reflect on the actual time-scales in the physics. A typical example is reacting flows where the time step is often guided by fast reactions, while the flow develops relatively slowly. The COMSOL Multiphysics software can therefore replace Δt^2 in the time-scale tensor with measures of type $(1/(\Delta \tilde{t})^2)^{-1}$, which are calculated from projections of weak expressions in a fashion similar to those in [Ref. 11](#). These measures of the time scale are used when **Use dynamic subgrid time scale** check box is selected. Streamline diffusion is active by default because it is necessary when convection is dominating the flow.

The governing equations for incompressible flow are subject to the Babuška–Brezzi condition, which states that the shape functions (basis functions) for pressure must be of lower order than the shape functions for velocity. If the incompressible Navier–Stokes equations are stabilized by streamline diffusion, it is possible to use equal-order interpolation. Hence, streamline diffusion is necessary when using first-order elements for both velocity and pressure. This applies also if the model is solved using geometric multigrid (either as a solver or as a preconditioner) and at least one multigrid hierarchy level uses linear Lagrange elements.

CROSSWIND DIFFUSION

Crosswind diffusion can also be formulated for systems of equations, and when applied to the Navier–Stokes equations it becomes a shock-capturing operator. COMSOL Multiphysics supports the formulation in [Ref. 9](#) with a shock-capturing viscosity of the Hughes–Mallet type [Ref. 8](#).

Incompressible flows do not contain shock waves, but crosswind diffusion is still useful for introducing extra diffusion in sharp boundary layers and shear layers that otherwise would require a very fine mesh to resolve.

Crosswind diffusion is active by default as it makes it easier to obtain a solution even if the problem is fully resolved by the mesh. Crosswind diffusion also enables the iterative solvers to use inexpensive presmoothers. If crosswind diffusion is deactivated, more expensive preconditioners must be used instead.

ISOTROPIC DIFFUSION

Isotropic diffusion adds diffusion to the Navier–Stokes equations. Isotropic diffusion significantly reduces the accuracy of the solution but does a very good job at reducing oscillations. The stability of the continuity equation is not improved.



[Numerical Stabilization](#) and [Iterative](#) in the *COMSOL Multiphysics Reference Manual*.

Solvers for Laminar Flow

The Navier–Stokes equations constitute a nonlinear equation system. A nonlinear solver must hence be applied to solve the problem. The nonlinear solver iterates to reach the final solution. In each iteration, a linearized version of the nonlinear system is solved using a linear solver. In the time-dependent case, a time marching method must also be applied. The default suggestions for each of these solver elements are discussed below.

NONLINEAR SOLVER

The nonlinear solver method depends on if the model solves a stationary or a time-dependent problem.

Stationary Solver

In the stationary case, a fully coupled, damped Newton method is applied. The initial damping factor is low since a full Newton step can be harmful unless the initial values are close to the final solution. The nonlinear solver algorithm automatically regulates the damping factor in order to reach a converged solution.

For advanced models, the automatically damped Newton method might not be robust enough. A pseudo time-stepping algorithm can then be invoked. See [Pseudo Time Stepping for Laminar Flow Models](#).

Time-Dependent Solver

In the time-dependent case, the initial guess for each time step is (loosely speaking) the previous time step, which is a very good initial value for the nonlinear solver. The automatic damping algorithm is then not necessary. The damping factor in the Newton method is instead set to a constant value slightly smaller than one. Also, for the same reason, it suffices to update the Jacobian once per time step.

It is seldom worth the extra computational cost to update the Jacobian more than once per time step. For most models it is more efficient to restrict the maximum time step or possibly lower the damping factor in the Newton method.

LINEAR SOLVER

The linearized Navier–Stokes equation system has saddle point character, unless the density depends on the pressure. This means that the Jacobian matrix has zeros on the diagonal. Even when the density depends on the pressure, the equation system effectively shares many numerical properties with a saddle point system.

For small 2D and 3D models, the default solver suggestion is a direct solver. Direct solvers can handle most nonsingular systems and are very robust and also very fast for small models. Unfortunately, they become slow for large models and their memory requirement scales as somewhere between $N^{1.5}$ and N^2 , where N is the number of degrees of freedom in the model. The default suggestion for large 2D and 3D models is therefore the iterative GMRES solver. The memory requirement for an iterative solver optimally scales as N .

GMRES is accelerated by a multigrid method, per default the smoothed aggregation algebraic multigrid (SAAMG) method. The cost of SAAMG is typically very low compared to the number of GMRES iterations necessary if no multigrid method is used. As the name implies, SAAMG builds its coarser meshes algebraically, so the application requires no additional meshes in order to employ SAAMG. In contrast, the geometric multigrid (GMG) method requires actual meshes. If a sufficient number of multigrid levels can be constructed, GMG is often faster than SAAMG. GMG is also superior for cluster computations and for shared-memory computations with many cores. When the default linear solver is GMRES, an optional, but deactivated, linear solver node is available where GMRES is accelerated by GMG.

Multigrid methods need smoothers, but the saddle point character of the linear system restricts the number of applicable smoothers. The choices are further restricted by the anisotropic meshes frequently encountered in fluid-flow problems. The efficiency of the smoothers is highly dependent on the numerical stabilization. Iterative solvers perform at their best when both [Streamline Diffusion](#) and [Crosswind Diffusion](#) are active.

The default smoother for P1+P1 elements is SCGS. This is an efficient and robust smoother specially designed to solve saddle point systems on meshes that contain anisotropic elements. The SCGS smoother works well even without crosswind diffusion. SCGS can sometimes work for higher-order elements, especially if **Method** in the SCGS settings is set to **Mesh element lines**. But there is no guarantee for this, so the

default smoother for higher order elements is an SOR Line smoother. SOR Line handles mesh anisotropy but does not formally address the saddle point character. It does, however, function in practice provided that streamline diffusion and crosswind diffusion are both active.

A different kind of saddle point character can arise if the equation system contains ODE variables. Some advanced boundary conditions can add equations with such variables. These variables must be treated with the Vanka algorithm. SCGS includes an option to invoke Vanka. Models with higher-order elements must apply SCGS or use the Vanka smoother. The latter is the default suggestion for higher-order elements, but it does not work optimally for anisotropic meshes.

TIME-DEPENDENT SOLVERS

The default time-dependent solver for Navier–Stokes is the BDF method with maximum order set to two. Higher BDF orders are not stable for transport problems in general nor for Navier–Stokes in particular.

BDF methods have been used for a long time and are known for their stability. However, they can have severe damping effects, especially the lower-order methods. Hence, if robustness is not an issue, a model can benefit from using the generalized- α method instead. Generalized- α is a solver which has properties similar to those of the second-order BDF solver but it is much less diffusive.

Both BDF and generalized- α are per default set to automatically adjust the time step. While this works well for many models, extra efficiency and accuracy can often be gained by specifying a maximum time step. It is also often beneficial to specify an initial time step to make the solver progress smoothly in the beginning of the time series.



In the *COMSOL Multiphysics Reference Manual*:

- [Time-Dependent Solver](#)
- [Multigrid, Direct, Iterative, SCGS, SOR Line, and Vanka](#)
- [Stationary Solver](#)

Pseudo Time Stepping for Laminar Flow Models

A stationary formulation has per definition no time derivatives and [Equation 3-11](#) reduces to:

$$\rho(\mathbf{u} \cdot \nabla)\mathbf{u} = \nabla \cdot [-p\mathbf{I} + \mu(\nabla\mathbf{u} + (\nabla\mathbf{u})^T)] + \mathbf{F} \quad (3-28)$$

Solving Equation 3-28 requires a starting guess that is close enough to the final solution. If no such guess is at hand, the fully transient problem can be solved instead. This is, however, a rather costly approach in terms of computational time. An intermediate approach is to add a fictitious time derivative to Equation 3-28:

$$\rho \frac{\mathbf{u} - \text{nojac}(\mathbf{u})}{\Delta \tilde{t}} + \rho(\mathbf{u} \cdot \nabla) \mathbf{u} = \nabla \cdot [-p \mathbf{I} + \mu(\nabla \mathbf{u} + (\nabla \mathbf{u})^T)] + \mathbf{F}$$

where $\Delta \tilde{t}$ is a *pseudo time step*. Since $\mathbf{u} - \text{nojac}(\mathbf{u})$ is always zero, this term does not affect the final solution. It does, however, affect the discrete equation system and effectively transforms a nonlinear iteration into a step of size $\Delta \tilde{t}$ of a time-dependent solver.

Pseudo time stepping is not active per default. The pseudo time step $\Delta \tilde{t}$ can be chosen individually for each element based on the local CFL number:

$$\Delta \tilde{t} = \text{CFL}_{\text{loc}} \frac{h}{|\mathbf{u}|}$$

where h is the mesh cell size. A small CFL number means a small time step. It is practical to start with a small CFL number and gradually increase it as the solution approaches steady state.

If the automatic expression for CFL_{loc} is set to the built-in variable CFLCMP , then the automatic setting suggests a PID regulator for the pseudo time step in the default solver. The PID regulator starts with a small CFL number and increases CFL_{loc} as the solution comes closer to convergence.

The default manual expression is

$$\begin{aligned} & 1.3^{\min(\text{niterCMP}, 9)} + \\ & \text{if}(\text{niterCMP} > 20, 9 \cdot 1.3^{\min(\text{niterCMP} - 20, 9)}, 0) + \\ & \text{if}(\text{niterCMP} > 40, 90 \cdot 1.3^{\min(\text{niterCMP} - 40, 9)}, 0) \end{aligned} \quad (3-29)$$

The variable `niterCMP` is the nonlinear iteration number. It is equal to one for the first nonlinear iteration. CFL_{loc} starts at 1.3 and increases by 30% each iteration until it reaches $1.3^9 \approx 10.6$. It remains there until iteration number 20 at which it starts to increase until it reaches approximately 106. A final increase after iteration number 40

then takes it to 1060. [Equation 3-29](#) can, for some advanced flows, increase CFL_{loc} too slowly or too quickly. CFL_{loc} can then be tuned for the specific application.



For details about the CFL regulator, see [Pseudo Time Stepping](#) in the *COMSOL Multiphysics Reference Manual*.

Discontinuous Galerkin Formulation

Some boundary conditions are implemented using a discontinuous Galerkin formulation. These boundary conditions include

- [Wall](#) — Slip, sliding walls, and moving walls (that is, walls with nonzero translational velocity).
- [Periodic Flow Condition](#)
- [Flow Continuity](#)

The formulation used in the Fluid Flow interfaces in COMSOL Multiphysics is the Symmetric Interior Penalty Galerkin method (SIPG). The SIPG method can be regarded to satisfy the boundary conditions in an integral sense rather than pointwise. More information on SIPG can be found in [Ref. 15](#).

In particular, the SIPG formulation includes a penalty parameter that must be large enough for the formulation to be coercive. The higher the value, the better the boundary condition is fulfilled, but a too high value results in an ill-conditioned equation system. The penalty parameter in COMSOL Multiphysics is implemented according to [Ref. 16](#).

Particle Tracing in Fluid Flow

The Particle Tracing Module is available to assist with these types of modeling problems.

It is possible to model particle tracing with COMSOL Multiphysics provided that the impact of the particles on the flow field is negligible. First compute the flow field, and then, as an analysis step, calculate the motion of the particles. The motion of a particle is defined by Newton's second law

$$m \frac{d^2 \mathbf{x}}{dt^2} = \mathbf{F}\left(t, \mathbf{x}, \frac{d\mathbf{x}}{dt}\right)$$

where \mathbf{x} is the position of the particle, m the particle mass, and \mathbf{F} is the sum of all forces acting on the particle. Examples of forces acting on a particle in a fluid are the drag force, the buoyancy force, and the gravity force. The drag force represents the force that a fluid exerts on a particle due to a difference in velocity between the fluid and the particle. It includes the viscous drag, the added mass, and the Basset history term. Several empirical expressions have been suggested for the drag force. One of those is the one proposed by Khan and Richardson (Ref. 13). That expression is valid for spherical particles for a wide range of particle Reynolds numbers. The particle Reynolds number is defined as

$$\text{Re}_p = \frac{|\mathbf{u} - \mathbf{u}_p| 2r\rho}{\mu}$$

where \mathbf{u} is the velocity of the fluid, \mathbf{u}_p the particle velocity, r the particle radius, ρ the fluid density, and μ the dynamic viscosity of the fluid. The empirical expression for the drag force according to Khan and Richardson is

$$\mathbf{F} = \pi r^2 \rho |\mathbf{u} - \mathbf{u}_p| (\mathbf{u} - \mathbf{u}_p) [1.84 \text{Re}_p^{-0.31} + 0.293 \text{Re}_p^{0.06}]^{3.45}$$




The model *Flow Past a Cylinder* (Application Library path **COMSOL_Multiphysics/Fluid_Dynamics/cylinder_flow**) demonstrates how to add and set up particle tracing in a plot group using the **Particle Tracing with Mass** node. It uses the predefined Khan-Richardson model for the drag force and neglects gravity and buoyancy forces.

References for the Single-Phase Flow, Laminar Flow Interfaces

1. G.G. Stokes, Trans. Camb. Phil. Soc., 8, pp. 287–305, 1845
2. P.M. Gresho and R.L. Sani, *Incompressible Flow and the Finite Element Method, Volume 2: Isothermal Laminar Flow*, John Wiley & Sons, 2000.
3. G.K. Batchelor, *An Introduction To Fluid Dynamics*, Cambridge University Press, 1967.
4. R.L. Panton, *Incompressible Flow*, 2nd ed., John Wiley & Sons, 1996.
5. I. Harari and T.J.R. Hughes, “What are C and h ? Inequalities for the Analysis and Design of Finite Element Methods”, *Comp. Meth. Appl. Mech. Engrg*, vol. 97, pp. 157–192, 1992.

6. Y. Bazilevs, V.M. Calo, T.E. Tezduyar, and T.J.R. Hughes, “ γ -Discontinuity Capturing for Advection-dominated Processes with Application to Arterial Drug Delivery”, *Int.J.Num. Meth. Fluids*, vol. 54, pp. 593–608, 2007.
7. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, 2nd ed., John Wiley & Sons, 2007.
8. T.J.R. Hughes and M. Mallet, “A New Finite Element Formulation for Computational Fluid Dynamics: III. The Generalized Streamline Operator for Multidimensional Advective-Diffusive System”, *Comp. Meth. Appl. Mech. Engrg*, vol. 58, pp. 305–328, 1986.
9. G. Hauke and T.J.R. Hughes, “A Unified Approach to Compressible and Incompressible Flows”, *Comp. Meth. Appl. Mech. Engrg*, vol. 113, pp. 389–395, 1994.
10. G. Hauke, “Simple Stabilizing Matrices for the Computation of Compressible Flows in Primitive Variables”, *Comp. Meth. Appl. Mech. Engrg*, vol. 190, pp. 6881–6893, 2001.
11. M.-C. Hsu, Y. Bazilevs, V.M. Calo, T.E. Tezduyar, and T.J.R. Hughes, “Improving stability of stabilized and multiscale formulations in flow simulations at small time steps”, *Comp. Meth. Appl. Mech. Engrg*, vol. 199, pp. 828–840, 2010.
12. D.J. Tritton, *Physical Fluid Dynamics*, 2nd ed., Oxford University Press, 1988.
13. J.M. Coulson and J.F. Richardson, “Particle Technology and Separation Processes”, *Chemical Engineering, Volume 2*, Butterworth-Heinemann, 2002.
14. J.L. Guermond, P. Mineev, and J. Shen, “An overview of projection methods for incompressible flows”, *Comp. Meth. Appl. Mech. Engrg*, vol. 195, pp. 6011–6045, 2006.
15. B. Rivière, *Discontinuous Galerkin Methods for Solving Elliptic and Parabolic Equations*, SIAM, 2008.
16. Y. Epshteyn and B. Rivière, “Estimation of penalty parameters for symmetric interior penalty Galerkin methods”, *J. Computational and Applied Mathematics*, vol. 206, pp. 843–872, 2007.
17. R.P. Chhabra and J.F. Richardson, *Non-Newtonian Flow and Applied Rheology*, 2nd ed., Elsevier, 2008.
18. Y. Bazilevs and T.J.R. Hughes, “Weak imposition of Dirichlet boundary conditions in fluid mechanics”, *Computers and Fluids*, vol. 36, pp. 12–26, 2007.

Porous Media and Subsurface Flow Interfaces

This chapter has information about the physics interfaces found under the **Fluid Flow>Porous Media and Subsurface Flow** branch (). It contains physics interfaces for the following sets of equations that apply under different conditions:

- [The Darcy's Law Interface](#) describes low-velocity flows of one or more fluids.
- [The Richards' Equation Interface](#) governs the wetting and drying of rocks or soils.
- [The Multiphase Flow in Porous Media Interface](#), which combines low velocity flow with phase transport of multiple immiscible phases in a porous medium is described in the [Multiphysics Interfaces and Couplings](#) chapter.
- [The Fracture Flow Interface](#) is a variant of Darcy's law that defines the flow along the interior boundaries representing fractures within a porous or solid medium.
- [The Layered Darcy's Law Interface](#) describes low-velocity flow of fluids in layered materials (like layered soils, paperboard, composites, etc.) represented by boundaries. It is only available in 3D models and is only active on those boundaries where a layered material is defined.
- [The Brinkman Equations Interface](#) is intended for porous media high-velocity flows.

- [The Free and Porous Media Flow Interface](#) combines free flow adjacent to porous domains.
- [The Laminar Flow and Creeping Flow Interfaces](#) use the *Navier-Stokes equations* (in which the inertia term is neglected in case of Creeping Flow) to describe free flows within a river or well, for example. The Laminar Flow interface is available with all COMSOL licenses.

With Darcy's law and Richards' equation, COMSOL Multiphysics solves for pressure and has physics interfaces for pressure head and hydraulic head. Set up these equations for single or multiple fluids. The physics interfaces can be coupled to other physics interfaces to describe poroelasticity, convection of heat- and density-driven flows, solute transfer, and electromagnetic forces, to name a few.

In this chapter you will find the following sections:

- [Theory for the Darcy's Law Interface](#)
- Darcy's Law:
 - [Theory for the Darcy's Law Interface](#)
 - [The Darcy's Law Interface](#)
- Richards' Equation:
 - [Theory for the Richards' Equation Interface](#)
 - [The Richards' Equation Interface](#)
- Fracture Flow:
 - [Theory for the Fracture Flow Interface](#)
 - [The Fracture Flow Interface](#)
- Layered Darcy's Law:
 - [Theory for the Layered Darcy's Law Interface](#)
 - [The Layered Darcy's Law Interface](#)
- Brinkman Equations:
 - [Theory for the Brinkman Equations Interface](#)
 - [The Brinkman Equations Interface](#)
- Free and Porous Media Flow:
 - [Theory for the Free and Porous Media Flow Interface](#)
 - [The Free and Porous Media Flow Interface](#)

Theory for the Darcy's Law Interface

The Darcy's Law Interface theory is described in this section:

- [About Darcy's Law](#)
- [Darcy's Law — Equation Formulation](#)
- [Storage Model](#)
- [Average Linear Velocity](#)
- [References for the Darcy's Law Interface](#)

About Darcy's Law

In a porous medium, the global transport of momentum by shear stresses in the fluid is often negligible, because the pore walls impede momentum transport to the fluid outside the individual pores. A detailed description, down to the resolution of every pore, is not practical in most applications. A homogenization of the porous and fluid media into a single medium is a common alternative approach. Darcy's law together with the continuity equation and equation of state for the pore fluid (or gas) provide a complete mathematical model suitable for a wide variety of applications involving porous media flows, for which the pressure gradient is the major driving force.

Darcy's law describes fluid movement through interstices in a porous medium. Because the fluid loses considerable energy to frictional resistance within pores, flow velocities in porous media are very low. The Darcy's Law interface can be applied to small-scale tasks in chemical engineering as well as to large-scale geophysical or hydrological tasks such as water moving in an aquifer or stream bank, oil migrating to a well, and even magma rising through the earth to a chamber in a volcano (see [Ref. 1](#), [Ref. 2](#), [Ref. 3](#), and [Ref. 4](#)). Also set up multiple Darcy's Law interfaces to model multiphase flows involving more than one mobile phase.

Darcy's law describes flow in porous media driven by gradients in the hydraulic potential field, which has units of pressure. For many applications it is convenient to represent the total hydraulic potential or the pressure and the gravitational components with equivalent heights of fluid or head. Division of potential by the fluid weight can simplify modeling because units of length make it straightforward to compare to many physical data. Consider, for example, fluid levels in wells, stream heights, topography, and velocities. The physics interface also supports specifying

boundary conditions and result evaluation using hydraulic head and pressure head. In the physics interface, pressure is always the dependent variable.

Darcy's Law — Equation Formulation

Darcy's law states that the velocity field is determined by the pressure gradient, the fluid viscosity, and the structure of the porous medium:

$$\mathbf{u} = -\frac{\kappa}{\mu} \nabla p \quad (4-1)$$

In this equation, \mathbf{u} is the Darcy's velocity or specific discharge vector (SI unit: m/s); κ is the permeability of the porous medium (SI unit: m²); μ is the fluid's dynamic viscosity (SI unit: Pa·s); p is the pore pressure (SI unit: Pa) and ρ is the density of the fluid (SI unit: kg/m³).



- The above equation formulation is called linear Darcy law. The applicability of this formulation is discussed in the chapter [About Darcian and Non-Darcian Flow](#) in the *Porous Media Flow Module User's Guide*.
- Permeability can be defined in different ways, either directly or based on predefined permeability models. These are described in detail in the chapter [Permeability Models](#).
- For a detailed overview of the permeability models available in each product visit <https://www.comsol.com/products/specifications/>.

The Darcy's Law interface combines Darcy's law with the continuity equation

$$\frac{\partial}{\partial t}(\rho \epsilon_p) + \nabla \cdot (\rho \mathbf{u}) = Q_m \quad (4-2)$$

In the above equation, ρ is the fluid density (SI unit: kg/m³), ϵ_p is the porosity, and Q_m is a mass source term (SI unit: kg/(m³·s)). Porosity is defined as the fraction of the control volume that is occupied by pores. Thus, the porosity can vary from zero for pure solid regions to unity for domains of free flow.

For large-scale applications it might be necessary to take gravity effects into account. Darcy's law then applies when the gradient in hydraulic potential drives fluid movement in the porous medium. Darcy's law then has the form:

$$\mathbf{u} = -\frac{\kappa}{\mu}(\nabla p + \rho \mathbf{g}) \quad (4-3)$$



- See the section [Gravity Effects](#) in the *Subsurface Flow Module User's Guide*.
- Note that gravity effects are only included in some modules. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.
- Gravity effects are not active by default. Select the check box **Include gravity** to activate the acceleration of gravity. Setting the elevation *D* to zero also turns off gravity effects.

Storage Model

Inserting Darcy's law ([Equation 4-1](#)) into the continuity equation produces the generalized governing equation

$$\frac{\partial}{\partial t}(\rho \epsilon_p) + \nabla \cdot \rho \left[-\frac{\kappa}{\mu}(\nabla p) \right] = Q_m \quad (4-4)$$

Represent this equation fully in COMSOL Multiphysics because relationships between density or permeability can be freely specified, for example, and pressure, temperature, concentration, and so on.

Expand the time-derivative term in [Equation 4-4](#)

$$\frac{\partial}{\partial t}(\rho \epsilon_p) = \epsilon_p \frac{\partial \rho}{\partial t} + \rho \frac{\partial \epsilon_p}{\partial t}$$

Define the porosity and the density as functions of the pressure, and apply the chain rule

$$\epsilon_p \frac{\partial \rho}{\partial t} + \rho \frac{\partial \epsilon_p}{\partial t} = \epsilon_p \frac{\partial \rho}{\partial p} \frac{\partial p}{\partial t} + \rho \frac{\partial \epsilon_p}{\partial p} \frac{\partial p}{\partial t}$$

Insert the definition of fluid compressibility $\chi_f = (1/\rho)(\partial \rho / \partial p)$ to the right-hand side and rearrange to arrive at

$$\frac{\partial(\rho \epsilon_p)}{\partial t} = \rho \left(\epsilon_p \chi_f + \frac{\partial \epsilon_p}{\partial p} \right) \frac{\partial p}{\partial t} = \rho S \frac{\partial p}{\partial t}$$

Using this relation, the generalized governing equation (Equation 4-4) takes the following form:

$$\rho S \frac{\partial p}{\partial t} + \nabla \cdot \rho \left[-\frac{\kappa}{\mu} (\nabla p) \right] = Q_m \quad (4-5)$$

In this equation, S is the storage coefficient (SI unit: 1/Pa), which can be interpreted as the weighted compressibility of the porous material and the fluid. The storage S can be an expression involving results from a solid-deformation equation or an expression involving temperatures and concentrations from other analyses. The Darcy's Law interface implements Equation 4-5 in the **Porous Medium** node, which explicitly includes different storage models to choose from, such as *linearized storage* which defines S (SI unit: 1/Pa) using the compressibility of fluids χ_f and of the porous matrix χ_p :

$$S = \varepsilon_p \chi_f + (1 - \varepsilon_p) \chi_p,$$

or the *poroelastic storage* where

$$S = \varepsilon_p \chi_f.$$



The Storage Model node is only available for the Subsurface Flow Module and the Porous Media Flow Module. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

Average Linear Velocity



Because fluids typically occupy only 10% to 50% of a porous medium, it follows that velocities within the pore channels exceed the Darcy velocity, \mathbf{u} , on the order of two to ten times. For clarity, the physics interface includes the average linear velocity within the pores, \mathbf{u}_α , (also termed the seepage velocity) defined as $\mathbf{u}_\alpha = \mathbf{u}/\varepsilon_p$, where ε_p is the porosity.

References for the Darcy's Law Interface

1. J. Bear, *Hydraulics of Groundwater*, McGraw-Hill, 1979.

2. S.E. Ingebritsen and W.E. Sanford, *Groundwater in Geologic Processes*, Cambridge University Press, 1998.
3. N.H. Sleep and K. Fujita, *Principles of Geophysics*, Blackwell Science, 1997.
4. D.L. Turcotte and G. Schubert, *Geodynamics*, Cambridge University Press, 2002.
5. J. Bear, *Dynamics of Fluids in Porous Media*, Elsevier Scientific Publishing, 1972.
6. Ö. Akgiray and A.M. Saatçı, “A New Look at Filter Backwash Hydraulics,” *Water Science and Technology: Water Supply*, vol. 1, no. 2, pp. 65–72, 2001.
7. L. Klinkenberg, “The permeability of porous media to liquids and gases,” *Drilling and Production Practice*, Am. Petroleum Inst., pp. 200–213, 1941.
8. Yu-Shu Wu and others, “Gas Flow in Porous Media with Klinkenberg Effects,” *Transport in Porous Media*, vol. 32, pp. 117–137, 1998.

The Darcy's Law Interface

The **Darcy's Law (dl)** interface (), found under the **Porous Media and Subsurface Flow** branch () when adding a physics interface, is used to simulate fluid flow through interstices in a porous medium. It can be used to model low-velocity flows or media where the permeability and porosity are very small, and for which the pressure gradient is the major driving force and the flow is mostly influenced by the frictional resistance within the pores. Set up multiple **Darcy's Law** interfaces to model multiphase flows involving more than one mobile phase. The Darcy's Law interface can be used for stationary and time-dependent analyses.

The main feature is the **Porous Medium** Properties node, which provides interfaces for defining the fluid material and the porous matrix properties.

When this physics interface is added, the following default nodes are also added in the **Model Builder — Porous Medium, No Flow** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and mass sources. You can also right-click **Darcy's Law** to select physics features from the context menu.

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 `dl`.

PHYSICAL MODEL

Enter a **Reference pressure level** p_{ref} (SI unit: Pa). The default value is `1[atm]`.

GRAVITY EFFECTS


By default there are no gravity effects added to Darcy's Law interface. Select the check box **Include gravity** to activate the acceleration of gravity. When this check box is selected a global **Gravity** feature node is shown in the interface model tree.

Enter a value for the acceleration of gravity. The default value `g_const` is the predefined standard acceleration of gravity on Earth.



This feature requires a specific license. For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>.

DISCRETIZATION

To display all settings available in this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.

You can choose the order of the shape functions used for the pressure variable solved by the Darcy's Law interface. The default shape functions are **Quadratic** Lagrange.

The **Compute boundary fluxes** check box is not activated by default. When this option is selected, the solver computes variables storing accurate boundary fluxes from each boundary into the adjacent domain.

If the check box is cleared, the COMSOL Multiphysics software instead computes the flux variables from the dependent variables using extrapolation, which is less accurate in postprocessing results but does not create extra dependent variables on the boundaries for the fluxes.



Also, the **Apply smoothing to boundary fluxes** check box is available if the previous check box is checked. The smoothing can provide a better behaved flux value close to singularities.

For details about the boundary fluxes settings, see [Computing Accurate Fluxes](#) in the *COMSOL Multiphysics Reference Manual*.

The **Value type when using splitting of complex variables** setting should in most pure mass transport problems be set to **Real** which is the default. It makes sure that the dependent variable does not get affected by small imaginary contributions, which can occur, for example, when combining a Time Dependent or Stationary study with a frequency-domain study. For more information, see [Splitting Complex-Valued Variables](#) in the *COMSOL Multiphysics Reference Manual*.

DEPENDENT VARIABLES

The dependent variable (field variable) is the **Pressure**. The name can be changed but the names of fields and dependent variables must be unique within a model.

	<ul style="list-style-type: none">• Domain, Boundary, Edge, Point, and Pair Nodes for the Darcy’s Law Interface• Theory for the Darcy’s Law Interface• Physical Constants in the <i>COMSOL Multiphysics Reference Manual</i>
	<ul style="list-style-type: none">• <i>Species Transport in the Gas Diffusion Layers of a PEM</i>: Application Library path Fuel_Cell_and_Electrolyzer_Module/Fuel_Cells/pem_gdl_species_transport_2d• <i>Steam Reformer</i>: Application Library path Chemical_Reaction_Engineering_Module/Reactors_with_Porous_Catalysts/steam_reformer• <i>Terzaghi Compaction</i>: Application Library path Subsurface_Flow_Module/Flow_and_Solid_Deformation/terzaghi_compaction

Domain, Boundary, Edge, Point, and Pair Nodes for the Darcy’s Law Interface

The [Darcy’s Law Interface](#) has the following domain, boundary, edge, point, and pair nodes, these 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).

DOMAIN

- Porous Medium
- Gravity
- Cross Section
- Initial Values
- Mass Source
- Thickness
- Porous Electrode Coupling
- Unsaturated Porous Medium

Note that some features are only available with certain COMSOL products (see <https://www.comsol.com/products/specifications/>).

BOUNDARY, EDGE, AND POINT

The following nodes (listed in alphabetical order) are available on exterior boundaries:

- Atmosphere/Gauge
- Electrode Surface Coupling
- Flux Discontinuity
- Fracture Flow
- Hydraulic Head
- Inlet
- Interior Wall
- Line Mass Source
- Mass Flux
- No Flow
- Outlet
- Pervious Layer
- Point Mass Source
- Precipitation
- Pressure
- Pressure Head
- Reaction Coefficients¹
- Symmetry
- Thin Barrier

¹ This node is described for the Transport of Diluted Species Interface

Note that some features are only available with certain COMSOL products (see <https://www.comsol.com/products/specifications/>)!




The relevant physics interface condition at interior boundaries is continuity:

$$\mathbf{n} \cdot (\rho_1 \mathbf{u}_1 - \rho_2 \mathbf{u}_2) = 0$$

The continuity boundary condition ensures that the pressure and mass flux are continuous. In addition, the [Pressure](#) boundary condition is available on interior boundaries.

The continuity boundary condition provides continuity for the pressure and mass flux. In addition, the following boundary conditions are available on interior boundaries:

- [Pressure](#)
 - [Pressure Head](#)
 - [Hydraulic Head](#)
 - [Interior Wall](#)
- [Pervious Layer](#)
 - [Flux Discontinuity](#)
 - [Fracture Flow](#)
 - [Thin Barrier](#)

	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.
	For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an Axial Symmetry node that is valid on the axial symmetry boundaries only.
	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.

Porous Medium

The **Porous Medium** node adds the equations for modeling flow through porous domains. By default these are [Equation 4-6](#) and [Equation 4-7](#) (excluding any mass sources).

$$\frac{\partial}{\partial t}(\varepsilon_p \rho) + \nabla \cdot (\rho \mathbf{u}) = Q_m \tag{4-6}$$

$$\mathbf{u} = -\frac{\kappa}{\mu} \nabla p \tag{4-7}$$

For a steady-state problem the first term in Equation 4-6 disappears. When Gravity is active Equation 4-7 is of the following form:

$$\mathbf{u} = -\frac{\kappa}{\mu}(\nabla p + \rho \mathbf{g}) \quad (4-8)$$

The **Porous Medium** feature has two default subfeatures — the **Fluid** subfeature where the fluid density ρ (SI unit: kg/m^3) and dynamic viscosity μ (SI unit: $\text{Pa}\cdot\text{s}$) are defined and the **Porous Matrix** subfeature where the permeability κ (SI unit: m^2) and porosity ϵ_p (dimensionless) are specified.

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 orthonormal coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The subnodes inherit these coordinate system settings. In particular, the **Permeability** or **Hydraulic conductivity** (in the **Porous Matrix** subnode) should be set according to the coordinate system selected in this section.

FLOW MODEL

Define if the flow follows a linear or nonlinear pressure-velocity relationship.

- **Darcian flow** (default) defines a linear relationship using Equation 4-6 and Equation 4-7.
- **Non-Darcian flow** defines a nonlinear relationship using Equation 4-6 and

$$\nabla p = -\frac{\mu}{\kappa}\mathbf{u} - \beta\rho|\mathbf{u}|\mathbf{u}$$

where the nonlinear parameter β (SI unit: $1/\text{m}$) is further specified in the **Porous Matrix** subfeature.

STORAGE MODEL

Equation 4-6 can also be formulated with respect to a storage term.

$$\rho S \frac{\partial p}{\partial t} + \nabla \cdot \rho \left[-\frac{\kappa}{\mu}(\nabla p) \right] = Q_m \quad (4-9)$$

The storage coefficient S can be interpreted as the weighted compressibility of the porous material and the fluid. Define the time dependent storage term by selecting one of the following options from the drop-down menu:

- **From density and porosity** (default), uses the formulation of [Equation 4-6](#)
- **Linearized storage**, where the following linear equation is used to define the storage:

$$S = \epsilon_p \chi_f + (1 - \epsilon_p) \chi_p$$

- **Poroelastic storage**, where $S = \epsilon_p \chi_f$
- **Quasistatic**, where the storage term is set to zero
- **User defined** to directly specify the storage coefficient S

Note that the option to choose a different **Flow model** and **Storage model** is only available in some modules. For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>.



- [Porous Material](#) in the *COMSOL Multiphysics Reference Manual*
- [About Darcian and Non-Darcian Flow](#) in the *Porous Media Flow Module User's Guide*
- [Storage Model](#) in the *Subsurface Flow Module User's Guide*

Fluid

This node defines the material properties of the fluid of the [Porous Medium](#) or [Unsaturated Porous Medium](#) parent node. The fluid can be specified as general gas or liquid or as an ideal gas.

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.

FLUID PROPERTIES

The available options for the **Fluid type** are **Gas/Liquid** (default) or **Ideal gas**. The selection determines how the density and compressibility of the fluid are set. The latter is only required if a poroelastic or linearized storage model has been selected in the parent [Porous Medium](#) node. For all fluid types specify the dynamic viscosity μ (SI unit: Pa·s).

Gas/Liquid

This option specifies the **Density** and the **Compressibility** for a general gas or liquid.

Ideal Gas

For **Ideal gas** it uses the ideal gas law to describe the fluid. In this case, specify the thermodynamics properties. Select a **Gas constant type** — **Specific gas constant** R_s (the default) or **Mean molar mass** M_n (SI unit: J/(mol·K)). For **Mean molar mass** the universal gas constant $R = 8.314$ J/(mol·K) is used as the built-in physical constant. For both properties, the defaults use values **From material**. For **User defined** enter another value or expression.

If required by a **Storage Model**, the **Compressibility** of the ideal gas is calculated as $\chi_f = 1/p_A$ with the absolute pressure p_A (Pa).

Porous Matrix

This node defines the porosity and material properties of the solid matrix of the [Porous Medium](#) or [Unsaturated Porous Medium](#) parent node.

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.

MATRIX PROPERTIES

The default **Porosity** ϵ_p of the solid matrix is taken **From material**. In a **Porous Material** node, the porosity is defined as follows:

$$\epsilon_p = 1 - \sum_i \theta_{si} - \sum_i \theta_{imfi}$$

where θ_{si} and θ_{imfi} are the porosities of the **Solid** and **Immobile Fluids** subnodes under the **Porous Material** node. See [Porous Material](#) in the *COMSOL Multiphysics Reference Manual* for more information.

Select **User defined** to enter a value or expression for ϵ_p .

If required by a **Storage model** in the parent node, specify the **Effective compressibility** of the porous matrix. The default value is $\chi_p = 10^{-4} Pa$.

Select a **Permeability model** to specify the capacity of the porous material to transmit flow. Which options are available depends on the **Flow model** setting in the parent node.

Note that for the **Unsaturated Porous Medium** parent node only the Darcian Flow models are available.

Darcian Flow

If **Darcian flow** is selected in the parent **Porous medium** node, select between the following options for the **Permeability model**:

- Select **Permeability** (default) to define the permeability of the porous matrix. The default **Permeability** κ (SI unit: m^2) uses the value **From material**, as defined by the **Porous material**. For **User defined** select **Isotropic** to define a scalar value or **Diagonal** or **Symmetric** to define a tensor value.
- Select **Hydraulic conductivity** or define a combination of fluid permeability and dynamic viscosity. For **Hydraulic conductivity** K (SI unit: m/s) select **Isotropic** to define a scalar value or **Diagonal** or **Symmetric** to define a tensor value. The default is $2.94 \cdot 10^{-4} \text{ m/s}$.
- Select **Kozeny-Carman** to define the permeability from the porosity and mean particle diameter. Enter the mean **Particle diameter** d_p (SI unit: m), the default value is 0.5 mm.

Non-Darcian Flow

Select between the following options for the **Permeability model**:

- For **Forchheimer**, the default **Permeability** κ (SI unit: m^2) uses the value **From material**, as defined by the **Porous material** list. Furthermore, the dimensionless **Forchheimer parameter** c_F can be defined. The default value is 0.55.
- If **Ergun** is selected, enter the mean **Particle diameter** d_p (default value: 0.5 mm).
- For **Burke-Plummer**, specify the mean **Particle diameter** d_p (default value: 0.5 mm).
- For **Klinkenberg**, the default **Permeability** κ_∞ (SI unit: m^2) uses the value **From material**. Enter the **Klinkenberg parameter** b_K . Its default value is $1 \text{e}3 \text{ Pa}$.

Note that some options are only available in some modules. For a detailed overview of the functionality available in each product, visit

<https://www.comsol.com/products/specifications/>.



• [About Darcian and Non-Darcian Flow](#) in the *Porous Media Flow Module User's Guide*

• [Permeability Models](#) in the *Porous Media Flow Module User's Guide*

RETENTION MODEL

For an [Unsaturated Porous Medium](#) select between the following options for the retention model:

- **van Genuchten**, to specify the retention model using the van Genuchten relationship. Enter the constitutive relation constants α (default is 1, and the SI unit is 1/m), n (default is 2), and l (default is 0.5). The constitutive parameter m is equal to $1 - 1/n$.
- **Brooks and Corey** to use the retention model according to the Brooks and Corey relationships.
- **User defined** to use a different retention model. Enter user defined expressions for the **Effective saturation** S_e , the **Liquid volume fraction** θ_l . The default is ϵ_p (the porosity variable). Specify the **Specific moisture capacity** C_m (SI unit: 1/m). The default is 0 (1/m). Enter an expression for the **Relative permeability** κ_r . The default is 1.

If there is a residual volume of liquid that cannot move through the pore network, also enter a value between 0 and 1 for the **Residual liquid volume fraction** θ_r . The default value is 0.



[Retention and Permeability Relationships](#) in the *Subsurface Flow Module User's Guide*

Mass Source

The **Mass Source** node adds a mass source Q_m , which appears on the right-hand side of the Darcy's Law equation ([Equation 4-6](#)).

$$\frac{\partial}{\partial t}(\rho\epsilon) + \nabla \cdot (\rho\mathbf{u}) = Q_m \quad (4-10)$$

For [The Fracture Flow Interface](#) — which is available for the Subsurface Flow Module and the Porous Media Flow Module — the mass source is applicable to the right-hand side of [Equation 4-22](#) (where it is multiplied with the fracture thickness).

MASS SOURCE

Enter a value or expression for the **Mass source** Q_m (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$). The default is $0 \text{ kg}/(\text{m}^3 \cdot \text{s})$.

Initial Values

The **Initial Values** node adds an initial value for the pressure that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

If you have the Subsurface Flow Module licensed, you can specify the pressure either directly or as an expression for the pressure head, H_p , or the hydraulic head, H ; the hydraulic head and the pressure head relate to the pressure p as

$$H_p = \frac{p}{\rho g}; \quad H = H_p + D$$

where ρ is the fluid density (SI unit: kg/m^3); g denotes the acceleration of gravity (SI unit: m/s^2); and D is the elevation (SI unit: m).

INITIAL VALUES

Enter a value or expression for the initial value of the **Pressure** p (SI unit: Pa). The default value is 0 Pa.

In case the Subsurface Flow Module is available, click the **Pressure head** button to enter a value or expression for H_p (SI unit: m). The default is 0 m. Click the **Hydraulic head** button to enter a value or expression for H (SI unit: m). The default is 0 m.

FRACTURE FLOW

For [The Fracture Flow Interface](#), enter a value or expression for the **Fracture thickness** d_f (SI unit: m). The default is 0.1 m.



Flow in a Fractured Reservoir: Application Library path

Subsurface_Flow_Module/Fluid_Flow/fractured_reservoir_flow

Unsaturated Porous Medium

The **Unsaturated Porous Medium** node adds the Richards' equation ([Equation 4-11](#)) for flow in variably saturated porous media.

$$\rho \left(\frac{C_m}{\rho g} + S_e S_p \right) \frac{\partial p}{\partial t} + \nabla \cdot \rho \left(-\frac{\kappa_r \kappa}{\mu} (\nabla p + \rho g \nabla D) \right) = Q_m \quad (4-11)$$

The Unsaturated Porous Medium feature has two default subfeatures — the [Fluid](#) subfeature where the fluid density ρ (SI unit: kg/m^3) and viscosity μ (SI unit: Pa·s) are

defined and the **Porous Matrix** subfeature where the permeability κ (SI unit: m^2) and porosity ε_p (dimensionless) are specified as well as the retention model.

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 orthonormal coordinate system (except boundary coordinate systems) added under the **Definitions** node.

The subnodes inherit these coordinate system settings. In particular, the **Permeability** or **Hydraulic conductivity** (in the **Porous Matrix** subnode) should be set according to the coordinate system selected in this section.

STORAGE MODEL

With this option, define the time storage term S_p in Equation 4-11.

- From density and porosity
- Linearized storage (default)
- User defined

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

Gravity

The **Gravity** node is automatically added when **Include gravity** is selected at interface level in the **Darcy's Law** settings and it is active in all domains in which the Darcy's Law interface is applied.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes. It can be used when prescribing the direction of the gravitational forces.

GRAVITY

Specify either the acceleration vector or — in case you have a Subsurface Flow Module license — the elevation.

When **Acceleration** is selected from the list, enter the components of the **Gravity vector** \mathbf{g} . The default value is $\mathbf{g_const}$ which is the physical constant having the value 9.8066 m/s^2 .

- For 3D and 2D axisymmetric models, the default value is $-\mathbf{g_const}$ in the z direction.
- For 2D models, the default value is $-\mathbf{g_const}$ in the y direction.

When **Elevation** is selected from the list, specify the **Elevation** D (SI unit: m). Select the check box **Specify reference position** to define a reference elevation.

Select the acceleration of gravity from either the Darcy's Law interface Settings (default), or specify a user defined value. When the check box **Include gravity** is not selected in the Darcy's Law interface Settings, the elevation D is set equal to zero.

Cross Section

Use this node with 1D components to model domains with another cross-sectional area than the global one that is used in the interface **Physical Model** section. In 1D geometries, the pressure is assumed to be constant in the radial direction, and Darcy's Law accounts for that.

CROSS SECTION

Enter values for the **Cross-sectional area** A_c to set the cross section of the domain in the plane perpendicular to the 1D geometry.

Thickness

Use this node with 2D and 1D axisymmetric components to model domains with another thickness than the overall thickness that is specified in the interface **Physical Model** section. In 2D geometries, the pressure is assumed to be constant in the out-of-plane direction (z direction with default spatial coordinate names). In 1D axisymmetric geometries the thickness represents the z direction.

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.



Porous Electrode Coupling

Use the **Porous Electrode Coupling** node to define a mass source based on the volumetric current densities of one or multiple **Porous Electrode Reaction** nodes in an Electrochemistry interface.

The source (or sink) is proportional to the **Molar mass (kg/mol)** of the reacting species, the current densities and the stoichiometric coefficients according to Faraday's law as defined by summation over the Reaction Coefficient subnodes.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

SPECIES

Based on the number of species required for the model, use the **Add** () and **Delete** () buttons under the table to add or remove **Species**. Then enter a value for the **Molar mass (kg/mol)** in the applicable rows for each species.

The setting in the table will determine the number of available fields for entering the stoichiometric coefficients of the Reaction Coefficient subnodes.



Electrode Surface Coupling

Use the **Electrode Surface Coupling** node to define a combined wall and inflow/outflow boundary condition based on current densities of one or multiple Electrode Reaction nodes in an Electrochemistry interface.

The flow is proportional to the **Molar mass (kg/mol)** of the reacting species, the current densities and the stoichiometric coefficients according to Faraday's law as defined by summation over the Reaction Coefficient subnodes.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

SPECIES

Based on the number of species required for the model, use the **Add** () and **Delete** () buttons under the table to add or remove **Species**. Then enter a value for the **Molar mass (kg/mol)** in the applicable rows for each species.

The setting in the table will determine the number of available field for entering the stoichiometric coefficients of the Reaction Coefficient subnodes.

Pressure


Use the **Pressure** node to specify the pressure on a boundary. In many cases the distribution of pressure is known, giving a Dirichlet condition $p = p_0$ where p_0 is a known pressure given as a number, a distribution, or an expression involving time, t , for example.

PRESSURE

Enter a value or expression for the **Pressure** p_0 (SI unit: Pa). Enter a relative pressure value in p_0 (SI unit: Pa).

For the Subsurface Flow Module, the **Pressure** node provides the pressure P_0 as a condition on edges in 3D models. Then select the edges under **Edge Selection**.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.

Pressure constraint on edges are available for use with the Fracture Flow boundary condition (which is available in the Subsurface Flow Module). This boundary condition needs constraints on the edges surrounding the fracture-flow boundary. In other cases, pressure constraints on edges are not implemented.

Mass Flux

Use the **Mass Flux** node to specify the mass flux into or out of the model domain through some of its boundaries. It is often possible to determine the mass flux from the pumping rate or from measurements. With this boundary condition, positive values correspond to flow into the model domain:

$$\mathbf{n} \cdot \rho \frac{\mathbf{K}}{\mu} (\nabla p + \rho g \nabla D) = N_0$$

where N_0 is a value or expression for the specified inward (or outward) Darcy flux. D is the elevation head which is set to zero for other than Subsurface Flow Module applications.

MASS FLUX

Enter a value or expression for the **Inward mass flux** N_0 . A positive value of N_0 represents an inward mass flux whereas a negative value represents an outward mass flux. The units are based on the geometric entity: **Boundaries**: (SI unit: $\text{kg}/(\text{m}^2 \cdot \text{s})$).

Line Mass Source

The **Line Mass Source** node adds mass flow originating from a tube of infinitely small radius.

SELECTION

The **Line Mass Source** feature is available for all dimensions, but the applicable selection differs between the dimensions.

MODEL DIMENSION	APPLICABLE GEOMETRICAL ENTITY
2D	Points
2D Axisymmetry	Points not on the symmetry axis
3D	Edges

LINE MASS SOURCE

Enter a value or expression for the source strength, N_0 (SI unit: kg/(m·s)). A positive value results in mass injection from the line into the computational domain, and a negative value means that the mass is removed from the computational domain.

Line sources located on a boundary affect the adjacent computational domains. This effect makes the physical strength of a line source located in a symmetry plane twice the given strength.

Point Mass Source

The **Point Mass Source** node models mass flow originating from an infinitely small sphere centered around a point. It is available for points in 3D geometries.

POINT MASS SOURCE

Enter a value or expression for the source strength, N_0 (SI unit: kg/(s)). A positive value results in mass injection from the point into the computational domain, and a negative value means that the mass is removed from the computational domain.

Point sources located on a boundary or on an edge affect the adjacent computational domains. This has the effect, for example, that the physical strength of a point source located in a symmetry plane is twice the given strength.

Inlet

The **Inlet** node adds a boundary condition for the inflow (or outflow) perpendicular (normal) to the boundary. It has three options that can be used to specify inlet condition on a boundary, as follows:

VELOCITY

Enter a value or expression for the **Normal inflow velocity** U_0 (SI unit: m/s). A positive value of U_0 represents an inflow velocity. A negative value represents an outflow velocity. The inlet velocity boundary condition is implemented as;

$$\mathbf{n} \cdot \rho \frac{\kappa}{\mu} (\nabla p + \rho g \nabla D) = \rho U_0$$

where U_0 is a value or expression for the specified inward (or outward) Darcy velocity. A positive value of the velocity U_0 corresponds to flow into the model domain whereas a negative value represents an outflow. D is the elevation head which is set to zero for other than Subsurface Flow Module applications.

PRESSURE

Use the **Pressure** option to specify the inlet pressure on a boundary. In many cases the distribution of pressure is known, giving a Dirichlet condition $p = p_0$ where p_0 is a known pressure given as a number, a distribution, or an expression involving time, t , for example. Enter a value or expression for the **Pressure** p_0 (SI unit: Pa).

MASS FLOW

If you select **Mass flow** as the inlet condition, specify the total **Mass flow rate** M_0 (SI unit: kg/s), or the **Pointwise mass flux** N_0 (SI unit: kg/(m²·s)).

With **Mass flow rate** boundary condition, positive values correspond to flow into the model domain:

$$-\int_{\partial\Omega} \rho(\mathbf{u} \cdot \mathbf{n}) dS = M_0$$


where M_0 is a value or expression for the specified inward (or outward) Darcy flux.

Pointwise mass flux boundary condition, positive values correspond to flow into the model domain:

$$\mathbf{n} \cdot \rho \frac{\kappa}{\mu} (\nabla p + \rho g \nabla D) = N_0$$

where N_0 is a value or expression for the specified inward (or outward) Darcy flux.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.

Symmetry

The **Symmetry** node describes a symmetry boundary. The following condition implements the symmetry condition on an axis or a flow divide:

$$\mathbf{n} \cdot \frac{\kappa}{\mu} \nabla p = 0$$

$$\mathbf{n} \cdot \frac{\kappa}{\mu} (\nabla p + \rho g \nabla D) = 0$$

For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node that is valid on the axial symmetry boundaries only.

No Flow

The **No Flow** node is the default boundary condition stating that there is no flow across impermeable boundaries. The mathematical formulation is:

$$\mathbf{n} \cdot \rho \frac{\kappa}{\mu} (\nabla p + \rho g \nabla D) = 0$$

where \mathbf{n} is the vector normal to the boundary. D is the elevation head which is set to zero for any other than Subsurface Flow Module applications.

Flux Discontinuity

Use the **Flux Discontinuity** node to specify a mass flux discontinuity through an interior boundary. The condition is represented by the following equation:

$$-\mathbf{n} \cdot (\rho \mathbf{u}_1 - \rho \mathbf{u}_2) = N_0$$

In this equation, \mathbf{n} is the vector normal (perpendicular) to the interior boundary, ρ is the fluid density, \mathbf{u}_1 and \mathbf{u}_2 are the Darcy velocities in the adjacent domains (as defined in [Equation 4-12](#)) and N_0 is a specified value or expression for the flux discontinuity.

$$\mathbf{u} = -\frac{\kappa}{\mu} \nabla p \quad (4-12)$$

For this boundary condition, a positive value of N_0 corresponds to a flow discontinuity in the opposite direction to the normal vector of the interior boundary.

MASS FLUX

Enter a value or expression for the **Inward mass flux** N_0 (SI unit: $\text{kg}/(\text{m}^2 \cdot \text{s})$). A positive value of N_0 represents a mass flux discontinuity in the opposite direction to the normal vector of the interior boundary.

Outlet

The **Outlet** node adds a boundary condition for the outflow (or inflow) perpendicular (normal) to the boundary. It has different options that can be used to specify inlet condition on a boundary, as follows:

VELOCITY

Enter a value or expression for the **Normal outflow velocity** U_0 (SI unit: m/s). A positive value of U_0 represents an outflow velocity whereas a negative value represents an inflow velocity.

$$-\mathbf{n} \cdot \rho \frac{\kappa}{\mu} \nabla p = \rho U_0$$

where U_0 is a specified value or expression for the outward (or inward) Darcy velocity.

PRESSURE

Similar to the inlet **Pressure** option, the outlet pressure on a boundary can be specified. Enter a value or expression for the **Pressure** p_0 (SI unit: Pa).

DISCHARGE

Enter a value or expression for the discharge Q_D to specify the volumetric flow rate through the boundary. The mass flux $\rho \mathbf{u}$ is related to the discharge as follows:

$$\int_{\partial\Omega} \rho \mathbf{u} \cdot \mathbf{ds} = \rho Q_D. \quad (4-13)$$

The mean normal outflow velocity at the boundary can then be calculated as $U_{\text{out}} = Q_D/A$, where A is the cross sectional area of the boundary.

Note that this option is only available for Porous Media Flow and Subsurface Flow modules. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

Precipitation

Enter a value or expression for the **Precipitation rate** which then contributes to the mass flux at the boundaries selected. If the boundary is inclined, a **Slope correction** can be applied.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

Interior Wall

The **Interior Wall** boundary condition can only be applied on interior boundaries. It is similar to the **No Flux** boundary available on exterior boundaries except that it applies on both sides of an interior boundary. It allows discontinuities of velocity and pressure across the boundary. The **Interior Wall** boundary condition can be used to avoid meshing thin structures by applying no-flux condition on interior curves and surfaces instead.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

Thin Barrier

The **Thin Barrier** boundary condition models interior permeable walls, membranes, geotextiles, or perforated plates as thin permeable barriers. The **Thin Barrier** boundary condition can only be applied on interior boundaries.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

WALL

Enter a value or expression for the **Thickness** d_b (SI unit: m, the default is 0.1 m) and for the **Permeability** κ_b (SI unit: m^2). The default **Permeability** κ_b uses the value **From material**. 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.

Pressure Head


Use the **Pressure Head** node to specify the pressure head (instead of the pressure) on a boundary. It adds this boundary condition for the pressure head $H_p = H_{p0}$, where H_{p0} is a known pressure head given as a number, a distribution, or an expression involving time, t , for example. The dimension of the pressure head is length (SI unit: m).

The Subsurface Flow Module license is required to use this boundary condition (see <https://www.comsol.com/products/specifications/>).

PRESSURE HEAD

Enter a value or expression for the **Pressure head** H_{p0} (SI unit: m). The default is 0 m.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.

Hydraulic Head


Use the **Hydraulic Head** node to specify the hydraulic head (instead of the pressure) on a boundary. This adds the Dirichlet condition for the hydraulic head $H = H_0$ where H_0 is a known hydraulic head given as a number, a distribution, or an expression involving time, t , for example.

The Subsurface Flow Module license is required to use this boundary condition (see <https://www.comsol.com/products/specifications/>).

HYDRAULIC HEAD

Enter a value or expression for the **Hydraulic head** H_0 (SI unit: m). The default is 0 m.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.


Atmosphere/Gauge

The **Atmosphere/Gauge** node specifies an atmospheric pressure or gauges the pressure to the atmospheric value. This means that the total hydraulic potential reduces to the gravitational pressure at the free surface. At a free surface, such as a spring or a seepage face, the pressure is atmospheric. If the pressures in the model is gauged to the

atmospheric value ($p = 0$), the total hydraulic potential reduces to the gravitational potential at the free surface — for example, the height of the free surface multiplied by the fluid weight, or $\rho_f g D$. This boundary condition sets the pressure at the boundary to zero and $p = 0$.

The Subsurface Flow Module license is required to use this boundary condition (see <https://www.comsol.com/products/specifications/>).

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.

Pervious Layer

The Subsurface Flow Module license is required to use this boundary condition (see <https://www.comsol.com/products/specifications/>).

The **Pervious Layer** node provides a boundary condition that describes a mass flux through a semi-pervious layer connected to an external fluid source at different pressure, pressure head, or hydraulic head. The model domain might connect to a larger body of water through the semi-pervious layer. This condition is represented with the following boundary condition:

$$\mathbf{n} \cdot \rho \frac{\kappa}{\mu} (\nabla p + \rho g \nabla D) = \rho R_b \left[\frac{(p_b - p)}{\rho g} + (D_b - D) \right] \quad (4-14)$$

In this equation, p_b (SI unit: Pa) and D_b (SI unit: m) are the pressure and the elevation of the distant fluid source, respectively, and R_b (SI unit: 1/s) is the conductance of materials between the source and the model domain (conductance to flow in the semi-pervious layer adjacent to the boundary). Typically $R_b = K'/B'$, where K' is the hydraulic conductivity (SI unit: m/s) of the layer and B' (SI unit: m) is its thickness. Using logical relationships, it is possible to activate these expressions at different times or under various flow conditions.

When the pressure head H_p is specified instead of the pressure, the boundary condition is the following:

$$\mathbf{n} \cdot \rho K \nabla (H_p + D) = \rho R_b [(H_{pb} - H_p) + (D_b - D)] \quad (4-15)$$

H_{pb} is the pressure head (SI unit: m) at the edge of the layer.

When the hydraulic head H is specified instead of the pressure head, the boundary condition becomes:

$$\mathbf{n} \cdot \rho K \nabla H = \rho R_b [H_b - H] \quad (4-16)$$

H_b is the hydraulic head (SI unit: m) at the edge of the layer.

K in Equation 4-15 and Equation 4-16 is the hydraulic conductivity.

At a free surface, such as a water table or seepage face, the pressure is atmospheric (here taken to be zero), so the total hydraulic potential equals gravitational potential, which is defined on D .



Gravity effects are not active by default. Select the check box **Include gravity** to activate the acceleration of gravity. Setting the elevation D to zero also turns off gravity effects.

PERVIOUS LAYER

Specify the material properties whether to specify an external pressure, pressure head, or hydraulic head.

Select an **External variable to specify** — **Pressure and elevation** to specify the external pressure, **Pressure head and elevation** to specify the external pressure head, or **Hydraulic head**. For all selections, enter a value for the **Conductance R_b** (SI unit: 1/s).

- For **Pressure and elevation** enter an **External pressure p_b** (SI unit: Pa) and **External elevation D_b** (SI unit: m).
- For **Pressure head and elevation** enter an **External pressure head H_{pb}** (SI unit: m), H_{pb} (SI unit: m) and **External elevation D_b** (SI unit: m).
- For **Hydraulic head** enter the **Hydraulic head H_b** (SI unit: m).

Well

The **Well** feature is intended to model injection or production wells and is available with the Subsurface Flow Module.

WELL

Enter a value or expression for the **Well diameter d_w** (SI unit: m, the default is 0.1 m). Select the Well type from the list, Production or Injection well.

Specify either the **Pressure** (SI unit: Pa), **Pressure head** (SI unit: m), **Hydraulic head** (SI unit: m), or the **Mass flow**. If you select **Mass flow**, specify the total **Mass flow rate** (SI unit: kg/s), the **Mass flow rate per unit length** (SI unit: kg/(m·s)) or the **Mass flux** (SI unit: kg/(m²·s)).

Fracture Flow

The **Fracture Flow** node adds fracture flow on boundaries using tangential derivatives to define the flow along interior boundaries representing fractures within a porous medium. It is implemented through [The Fracture Flow Interface](#). Additional subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

Theory for the Richards' Equation Interface

The [Richards' Equation Interface](#) theory is described in this section:

- [About Richards' Equation](#)
- [Retention and Permeability Relationships](#)
- [References for the Richards' Equation Interface](#)

About Richards' Equation

Richards' equation models flow in variably saturated porous media. With variably saturated flow, hydraulic properties change as fluids move through the medium, filling some pores and draining others.

This discussion of the Richards' Equation interface begins with the propagation of a single liquid (oil or water). The pore space not filled with liquid contains an immobile fluid (air) at atmospheric pressure.

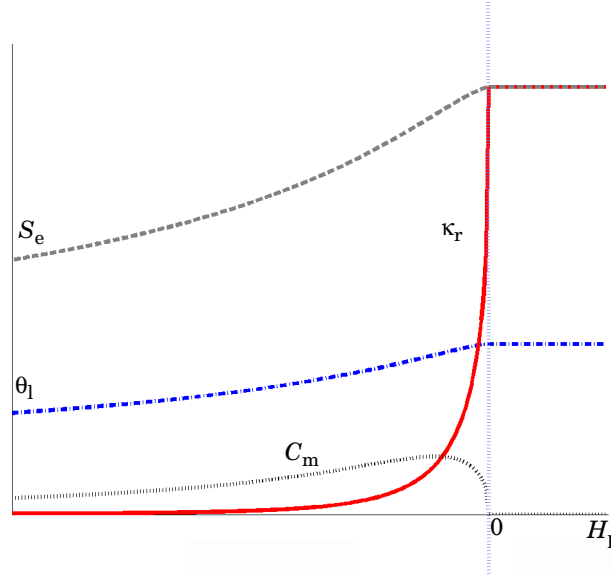


Figure 4-1: Fluid retention and permeability functions that vary with pressure head, as given by Van Genuchten formulas available in the Richards' Equation interface.

Many efforts to simplify and improve the modeling of flow in variably saturated media have produced a number of variants of Richards' equation since its appearance. The form that COMSOL Multiphysics solves is very general and allows for time-dependent changes in both saturated and unsaturated conditions (see [Ref. 1](#) and [Ref. 2](#)):

$$\rho \left(\frac{C_m}{\rho g} + S_e S_p \right) \frac{\partial p}{\partial t} + \nabla \cdot \rho \left(-\frac{\kappa_r \kappa_s}{\mu} (\nabla p + \rho g \nabla D) \right) = Q_m \quad (4-17)$$

where the pressure, p , is the dependent variable. In this equation, C_m represents the specific moisture capacity, S_e denotes the effective saturation, S_p is the storage coefficient, κ_s gives the hydraulic permeability at saturation, μ is the fluid dynamic viscosity, κ_r denotes the relative permeability, ρ is the fluid density, g is acceleration of gravity, D represents the elevation, and Q_m is the fluid source (positive) or sink (negative).

The fluid velocity across the faces of an infinitesimally small surface is

$$\mathbf{u} = -\frac{\kappa_r \kappa_s}{\mu} (\nabla p + \rho g \nabla D)$$

where \mathbf{u} is the velocity vector. The porous medium consists of pore space, fluids, and solids, but only the liquids move. The equation above describes the flux as distributed across a representative surface. To characterize the fluid velocity in the pores, COMSOL Multiphysics also divides \mathbf{u} by the liquid volume fraction, θ_l . This interstitial, pore or average linear velocity is $\mathbf{u}_a = \mathbf{u}/\theta_l$.

Retention and Permeability Relationships

Richards' equation appears deceptively similar to the saturated flow equation set out in the Darcy's Law interface, but it is notoriously nonlinear (Ref. 3). Nonlinearities arise because the material and hydraulic properties θ_l , S_e , C_m , and κ_r vary for unsaturated conditions (for example, negative pressure) and reach a constant value at saturation (for example, pressure of zero or above). The volume of liquid per porous medium volume, θ_l , ranges from a small residual value θ_r to the total porosity ϵ_p . Its value is given in a constitutive relation in the model commonly defined as:

$$\theta_l = \theta_r + S_e(\theta_s - \theta_r) \quad (4-18)$$

The variable θ_s is the saturated liquid volume fraction and therefore $\theta_s = \epsilon_p$. The effective saturation, S_e has a maximum value of 1 at saturation and usually follows a nonlinear relationship if the porous medium is unsaturated. The specific moisture capacity, C_m , describes the change in θ_l as the solution progresses, the slope on a plot of θ_l versus pressure (or pressure head). The relative permeability, κ_r , increases with moisture content and varies from a nominal value to 1 at saturation, which reveals that the fluid moves more readily when the porous medium is fully wet.

The Richards' Equation interface in COMSOL Multiphysics includes three retention models to define θ_l , S_e , C_m , and κ_r :

- The analytic formulas of van Genuchten.
- The analytic formulas of Brooks and Corey.
- User-defined expression.

The analytic formulas of van Genuchten (Ref. 4) and Brooks and Corey (Ref. 5) are so frequently used that they are synonymous with this variably saturated flow modeling. Posed in terms of pressure head $H_p = p/(\rho g)$, the analytic expressions require data for the saturated θ_s and residual θ_r liquid volume fractions as well as constants α , n , m , and l , which specify a particular medium type.

VAN GENUCHTEN

The van Genuchten equations define saturation when the fluid pressure is atmospheric (that is, $H_p = 0$). For unsaturated conditions ($H_p < 0$) the effective saturation is defined as:

$$S_e = \begin{cases} \frac{1}{1 + |\alpha H_p|^{nm}} & H_p < 0 \\ 1 & H_p \geq 0 \end{cases}$$

For the liquid volume fraction [Equation 4-18](#) is used. The relative permeability is defined as

$$\kappa_r = S_e^l \left[1 - \left(1 - S_e^{\frac{1}{m}} \right)^m \right]^2$$

which results in $\kappa_r = 1$ at saturation ($S_e = 1$). The specific moisture capacity is defined as:

$$C_m = \frac{\alpha m}{1 - m} (\theta_s - \theta_r) S_e^{\frac{1}{m}} \left(1 - S_e^{\frac{1}{m}} \right)^m$$

here, the constitutive parameter m is equal to $1 - 1/n$. The functions are highly nonlinear which is also depicted in [Figure 4-2](#)

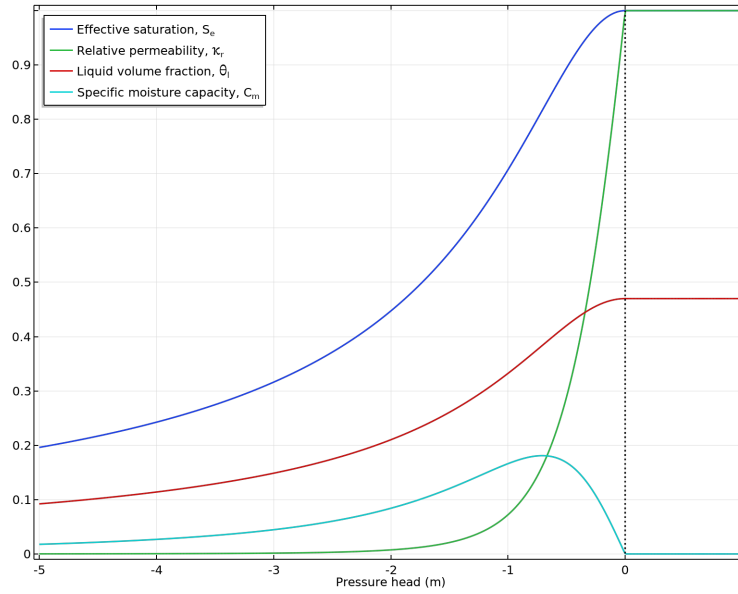


Figure 4-2: Van Genuchten retention functions for $\alpha = 1 \text{ m}^{-1}$, $n = 2$, and $l = 0.5$.

BROOKS AND COREY

With the Brooks and Corey approach, an air-entry pressure distinguishes saturated ($H_p > -1/\alpha$) and unsaturated ($H_p < -1/\alpha$) flow so that

$$S_e = \begin{cases} \frac{1}{|\alpha H_p|^n} & H_p < -\frac{1}{\alpha} \\ 1 & -\frac{1}{\alpha} \leq H_p \end{cases}$$

Again [Equation 4-18](#) describes the liquid volume fraction. The relative permeability is defined as

$$\kappa_r = S_e^{\frac{2}{n} + l + 2}$$

which results in $\kappa_r = 1$ at saturation ($S_e = 1$). The specific moisture capacity is defined as:

$$C_m = \frac{-n}{H_p} (\theta_s - \theta_r) \frac{1}{S_e}$$

Here, the constitutive parameter m is equal to $1 - 1/n$. The functions are highly nonlinear which is also depicted in [Figure 4-3](#)

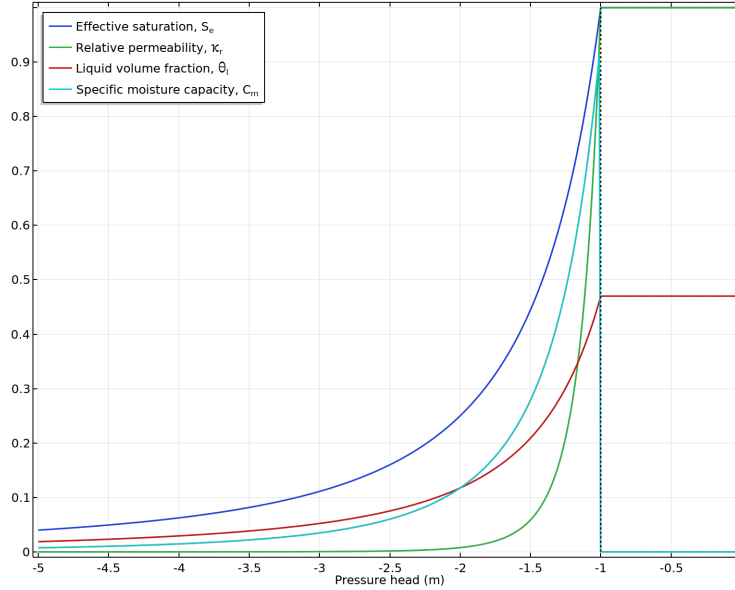


Figure 4-3: The Brooks and Corey retention functions for $\alpha = 1 \text{ m}^{-1}$, $n = 2$, and $l = 0.5$.

USER DEFINED



COMSOL Multiphysics also provides user-defined options for those who want to incorporate experimental data or arbitrary expressions to define these relationships. Enter expressions for θ_l , S_e , C_m , and κ_r directly.

References for the Richards' Equation Interface

1. J. Bear, *Dynamics of Fluids in Porous Media*, Elsevier Scientific Publishing, 1972.
2. J. Bear, *Hydraulics of Groundwater*, McGraw-Hill, 1979.
3. R.A. Freeze, "Three-dimensional, Transient, Saturated-unsaturated Flow in a Groundwater Basin," *Water Resour.Res.*, vol. 7, no. 2, 1971.

4. M.Th. van Genuchten, "A Closed-form Equation for Predicting the Hydraulic Conductivity of Unsaturated Soils," *Soil Sci. Soc. Am. J.*, vol. 44, 1980.
5. 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 Richards' Equation Interface

The **Richards' Equation (dl)** interface (), found under the **Porous Media and Subsurface Flow** branch () when adding a physics interface, is used to analyze flow in variably saturated porous media. The analytic formulas of van Genuchten, Brooks, and Corey are available for modeling variably saturated flow, where hydraulic properties change as fluids move through the porous medium, filling some pores and draining others. The physics interface can be used for stationary and time-dependent analysis.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Richards' Equation: Unsaturated Porous Medium** (which adds the equation for the pressure and provides an interface for defining the fluid material and its properties, including the retention model), **No Flow** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and mass sources. You can also right-click **Richards' Equation** to select physics features from the context menu.

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 `dl`.

PHYSICAL MODEL

Enter a **Reference pressure level** p_{ref} (SI unit: Pa). The default value is `1[atm]`.

GRAVITY EFFECTS

Gravity effects are active by default.eselect the check box **Include gravity** to deactivate the acceleration of gravity.


Under **Acceleration of gravity**, enter the value for the acceleration of gravity (SI unit m/s^2). The default value is taken from the predefined physical constant `g_const`, which is the standard acceleration of gravity on earth.



DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Pressure**. The name can be changed but the names of fields and dependent variables must be unique within a model.

DISCRETIZATION



You can choose the order of the shape functions used for the pressure variable solved by the Richards' Equation interface. The default shape functions are **Quadratic** Lagrange.

To display all settings available in this section, click the **Show** button () and select **Advanced Physics Options..**

	<ul style="list-style-type: none">• Theory for the Richards' Equation Interface• Domain, Boundary, Edge, Point, and Pair Nodes for the Richards' Equation Interface• Physical Constants in the <i>COMSOL Multiphysics Reference Manual</i>
	<ul style="list-style-type: none">• <i>Variably Saturated Flow</i>: Application Library path Subsurface_Flow_Module/Fluid_Flow/variably_saturated_flow• <i>Pesticide Transport and Reaction in Soil</i>: Application Library path Subsurface_Flow_Module/Solute_Transport/pesticide_transport

Domain, Boundary, Edge, Point, and Pair Nodes for the Richards' Equation Interface

The [Richards' Equation Interface](#) has the same domain, boundary, edge, point, and pair nodes described for the [The Darcy's Law Interface](#).

	Domain, Boundary, Edge, Point, and Pair Nodes for the Darcy's Law Interface
	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.



For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node that is valid on the axial symmetry boundaries only.



In the *COMSOL Multiphysics Reference Manual* see [Table 2-4](#) for links to common sections and [Table 2-5](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Flownet Plot

The Flownet plot is available for 2D and 2D axisymmetric models. This plot is created automatically when solving a model that includes a Richards' equation interface, and it is created by adding a Streamline plot for the velocity field and a contour plot for the pressure.

Theory for the Fracture Flow Interface

The [Fracture Flow Interface](#) theory is described in this section:

- [About Fracture Flow](#)
- [Equations and Inputs](#)

About Fracture Flow

The Fracture Flow interface uses tangential derivatives to define the flow along the interior boundaries representing fractures within a porous (or solid) block. Use this physics interface on boundaries in 3D, 2D, and axisymmetric geometries.



Fracture Flow is also available as a boundary condition in the Darcy's Law and Richards' Equation interfaces.

The physics interface uses the tangential version of Darcy's law:

$$\mathbf{q}_f = -\frac{\kappa_f}{\mu} d_f (\nabla_T p + \rho g \nabla_T D) \quad (4-19)$$

Here, \mathbf{q}_f is the volume flow rate per unit length in the fracture, κ_f is the fracture's permeability, μ is the fluid dynamic viscosity, d_f is the *aperture* or fracture thickness, ∇_T denotes the gradient operator restricted to the fracture's tangential plane, p is the pressure, ρ is the fluid density, g is the acceleration of gravity, and D represents the vertical coordinate.



Gravity effects are not active by default. Select the check box **Include gravity** to activate the acceleration of gravity. Setting the elevation D to zero also turns off gravity effects.

The variable \mathbf{q}_f gives the volume flow rate per unit length of the fracture. The mean fluid velocity within the fracture is $\mathbf{u}_f = \mathbf{q}_f / d_f$.

It is also possible to use the hydraulic conductivity of the fracture, K_f (SI unit: m/s), to define the capacity to transmit flow instead of using the fracture's permeability κ_f and fluid viscosity μ . These quantities are related by

$$\frac{\kappa_f}{\mu} = \frac{K_f}{\rho g}$$

The hydraulic conductivity represents properties of both fluid and porous matrix. If the model is defined using the hydraulic conductivity, Equation 4-19 changes to

$$\mathbf{q}_f = -\frac{K_f}{\rho g} d_f (\nabla_T p + \rho g \nabla_T D) \quad (4-20)$$

The Cubic law equation describes the permeability of the fracture from the aperture or fracture's thickness d_f and the roughness factor f_f

$$\kappa_f = \frac{d_f^2}{12 f_f}$$

If the model is defined using the Cubic law equation, the expression for flow rate per unit length in the fracture in Equation 4-19 changes to

$$\mathbf{q}_f = -\frac{d_f^3}{12 \mu f_f} (\nabla_T p + \rho g \nabla_T D) \quad (4-21)$$

Together with the material properties, Equation 4-19 above, in combination with the continuity equation integrated over the fracture cross section, produces a single equation for the pressure.

$$d_f \frac{\partial}{\partial t} (\epsilon_f \rho) + \nabla_T \cdot (\rho \mathbf{q}_f) = d_f Q_m \quad (4-22)$$

where ϵ_f is the fracture porosity, and Q_m is the mass source term (SI unit: kg/(m³·s)). The aperture or fracture thickness d_f can vary along the fracture and therefore appears on both sides of the equation.

The physics interface solves for the same dependent variable as for the equation in the porous medium, the pressure p .

Equations and Inputs

Two material models are supported: Fluid and Fracture Properties and the Storage Model.

FLUID AND FRACTURE PROPERTIES

This is the default setting of the Fracture Flow interface, and implements the fracture flow as described in [Equation 4-22](#). The values for the fluid properties (density and dynamic viscosity) can be taken from the material node or be defined by an arbitrary expression. The same can be done for the fracture's properties, such as porosity and permeability. Enter the fracture's thickness in the corresponding field.

STORAGE MODEL



The governing equation takes the slightly different form, compared to [Equation 4-22](#):

$$\rho S_f d_f \frac{\partial p}{\partial t} - \nabla_T \cdot (\rho \mathbf{q}_f) = d_f Q_m$$

where S_f is the fracture storage coefficient (SI unit: 1/Pa).

In contrast to the Darcy's Law and Richards' Equation interfaces, the storage coefficient is always a user-defined expression in the Fracture Flow interface.

The Fracture Flow Interface

The **Fracture Flow (esff)** interface (), found under the **Porous Media and Subsurface Flow** branch () when adding a physics interface, is used to simulate flow along interior boundaries representing fractures within a porous or solid medium. The physics interface uses a variant of Darcy's law and can be used for stationary and time-dependent analysis.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Fluid and Fracture Properties**, **No Flow** (the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, mass sources and conditions on edges or points. You can also right-click **Fracture Flow** to select physics features from the context menu.

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 `esff`.

PHYSICAL MODEL

Enter a **Reference pressure level** p_{ref} (SI unit: Pa). The default value is `1[atm]`.

GRAVITY EFFECTS

Gravity effects are not active by default. Select the check box **Include gravity** to activate the acceleration of gravity.


Under **Acceleration of gravity** enter the value for the acceleration of gravity g (SI unit: m/s^2). The default value is the predefined constant `g_const`.

DISCRETIZATION

You can choose the order of the shape functions used for the pressure variable solved by the Fracture Flow interface. The default shape functions are **Quadratic** Lagrange.

DEPENDENT VARIABLES

The dependent variable (field variable) is for the **Pressure**. The name can be changed but the names of fields and dependent variables must be unique within a model.

To display all settings available in this section, click the **Show** button () and select **Advanced Physics Options**.



- [Domain, Boundary, Edge, Point, and Pair Nodes for the Fracture Flow Interface](#)
- [Theory for the Fracture Flow Interface](#)
- [Physical Constants](#) in the *COMSOL Multiphysics Reference Manual*

Domain, Boundary, Edge, Point, and Pair Nodes for the Fracture Flow Interface

The [Fracture Flow Interface](#) has the domain, boundary, edge, point, and pair nodes 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.

DOMAIN

- [Fluid and Fracture Properties](#)
- [Aperture](#)
- [Initial Values](#)
- [Mass Source](#)
- [Storage Model](#)
- [Gravity](#)
- [Thickness](#)



Storage Model and **Mass Source** are described for the Darcy’s Law interface and are applied to boundaries instead of domains.

BOUNDARY CONDITIONS

The boundary conditions for this physics interface are similar to the boundary conditions for [The Darcy’s Law Interface](#) but are applied to edges in 3D and points in 2D. Edges and points form the “boundaries” of the geometries where the **Fracture Flow** interface is defined, which are faces in 3D and boundaries in 2D.



In the descriptions for these nodes, replace the **Boundary Selection** section with the **Edge Selection** (3D) or **Point Selection** (2D) section when using the Fracture Flow interface.

The following nodes are available on exterior boundaries (edges and points) and described for [The Darcy’s Law Interface](#):

- [Pressure](#)
- [Pressure Head](#)
- [Hydraulic Head](#)
- [Inlet](#)
- [Outlet](#)
- [Mass Flux](#)
- [Symmetry](#)
- [No Flow](#) (the default)
- [Atmosphere/Gauge](#)
- [Pervious Layer](#)

Note that some of the boundary conditions mentioned above are only available for certain modules. For detailed information, see <https://www.comsol.com/products/specifications/>.

The relevant physics interface condition at interior edges is continuity. In addition, the following edge conditions are available on interior edges:

- [Pressure](#)
- [Pervious Layer](#)



In the *COMSOL Multiphysics Reference Manual* see [Table 2-4](#) for links to common sections and [Table 2-5](#) to common feature nodes. You can also search for information; press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Fluid and Fracture Properties

The **Fluid and Fracture Properties** node adds Equation 4-2 (excluding any mass sources) on the selected boundary for the pressure and provides an interface for defining the fluid material and its properties, including the effective porosity.



All the nodes are the same as with the Darcy's Law interface except where indicated.

FLUID PROPERTIES

The default **Fluid material** uses the **Domain material** (the material defined for the domain). Select another material as needed.

The default **Density** ρ (SI unit: kg/m^3) uses values **From material** based on **Fluid material** selected. For **User defined** enter another value or expression. The default is $0 \text{ kg}/\text{m}^3$.

The default **Dynamic viscosity** μ (SI unit: $\text{Pa}\cdot\text{s}$) uses values **From material** based on the **Fluid material** selected. For **User defined** enter another value or expression. The default is $0 \text{ Pa}\cdot\text{s}$.



If **Hydraulic conductivity** is selected as the **Permeability model** (see the **Matrix Properties** section), dynamic viscosity is not available. The hydraulic conductivity is defined using a combination of fluid and matrix properties and replaces the need of defining the dynamic viscosity.

FRACTURE PROPERTIES

The default **Porous material** uses the **Boundary material** (the material defined for the fracture domain) for the fracture. Select another material as needed.

Porosity

The default **Porosity** in the fracture ϵ_f (a dimensionless number between 0 and 1) uses values **From material** based on the **Porous material** selected. For **User defined** enter another value or expression. The default is 0.

Permeability Model

Select a **Permeability model** to specify the capacity of the porous material to transmit flow: **Permeability** to define the permeability of the fracture, **Hydraulic conductivity**,

which is a combination of permeability and the dynamic viscosity of water, or **Cubic law** to define the permeability as a function of the fracture's aperture.

- For **Permeability** κ_f (SI unit: m^2) the default uses values **From material** as defined by the **Porous material** selected. For **User defined** select **Isotropic** to define a scalar value or **Diagonal**, **Symmetric**, or **Full** to enter a tensor value.
- For **Hydraulic conductivity** K (SI unit: m/s) enter a value or expression and select **Isotropic** to define a scalar value or **Diagonal**, **Symmetric**, or **Full** to enter a tensor value.
- For **Cubic law** enter a value or expression for the **Roughness factor** f_f .

Storage Model

The settings are the same as for the [Fluid and Fracture Properties](#). Enter a value or expression for Storage S (SI unit: $1/\text{Pa}$).

Aperture

The default node to specify the fracture's aperture. Enter a value or expression for the **Aperture** d_f (SI unit: m). The default is 1 mm.

Initial Values

The **Initial Values** node adds an initial value for the pressure, pressure head, or hydraulic head pressure that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

- Select the **Pressure** (the default) button to enter a value or expression for the initial value of the pressure p (SI unit: Pa). The default is 0 Pa.
- Select **Pressure head** to enter a value or expression for H_p (SI unit: m).
- Select **Hydraulic head** to enter a value or expression for H (SI unit: m).

Theory for the Layered Darcy's Law Interface

The Theory for the Layered Darcy's Law interface is based on Darcy's Law which is described in [Theory for the Darcy's Law Interface](#). Therefore, this section focuses on the 2D approximation for layered materials.

The Darcy's Law with continuity equation can be found in Section [Theory for the Darcy's Law Interface](#), Equation 4-2. The Darcy velocity can be written as:

$$\mathbf{u} = -\frac{\kappa}{\mu}(\nabla p - \rho g \nabla D) \quad (4-23)$$

where κ is the permeability of porous material, μ is the dynamic viscosity of fluid, ρ the fluid density, p is the fluid pressure, g is the gravitational constant, and D is the elevation.

Gradient Handling

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 p as:

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

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 p as:

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

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

$$\nabla p = \nabla_{\mathbf{t}} p + \nabla_{\mathbf{n}} p. \quad (4-24)$$

[Equation 4-24](#) is valid for flat layered shells. However, for curved shells, 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:

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

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, l_{sc} , 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

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

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.

Then, the gradient becomes

$$\nabla p = \left[\left(\frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} \right)^{-1} \right]^T \cdot \left(\nabla_{\mathbf{t}} p + \frac{d \text{tang}(p, x_d)}{l_{sc}} \cdot \mathbf{n}^T \right)$$

where x_d is the extra dimension coordinate, \mathbf{n} is the positive normal direction, and

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

where

- \mathbf{X}_r are the reference surface coordinates
- z is the extradimension thickness coordinate (varying between 0 and d)
- z_{off} is the relative midplane offset
- d is the layered material thickness.

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



$$\text{ASF} = \left| \frac{\partial \mathbf{X}}{\partial \mathbf{X}_r} \right|$$

Related information:



- [Modeling Layered Materials](#) in the *Heat Transfer Module User's Guide*
 - [Results Evaluation in Layered Shells](#) in the *Structural Mechanics User's Guide*
 - 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*.
-

The Layered Darcy's Law Interface

The **Layered Darcy's Law (ldl)** interface () , found under the **Porous Media and Subsurface Flow** branch () when adding a physics interface, is used to simulate flow within layered porous materials which are defined on boundaries within a 3D model domain. It is only available for 3D simulations and is only active on all boundaries where a layered material is defined.



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.

The physics interface uses a variant of Darcy's law and can be used for stationary and time-dependent analysis. Although the layered material is represented as a boundary in the model, the through-thickness variation of the pressure is accounted for. In addition, a single boundary may represent several layers with different 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.

When this physics interface is added, the following default nodes are also added to the **Model Builder: Fluid and Matrix Properties, No Flow** (the default edge condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, edge conditions and mass sources. You can also right-click **Layered Darcy's Law** to select physics features from the context menu.

Settings for the Layered Darcy's Law 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 `ld1`.

SHELL PROPERTIES

Here you select specific layers in a layered material on which the physics interface should be active. By default, the **Use all layers** check box is selected. This means that all layers in all layered materials on the selected boundaries are used.

If you clear the **Use all layers** check box, you can select individual layers within a single layered material. This means that the physics interface is restricted to the boundaries on which a specific layered material is defined.



For a general description of layer selections, see [The Shell Properties and Interface Selection Sections](#) in the *Structural Mechanics Module User's Guide*.

PHYSICS SYMBOLS

There are physics symbols available with some physics features to help you to graphically indicate boundary conditions, boundary normal orientation, loads, and other features. To display them select the **Enable Physics Symbols** check box under **Physics Symbols** in the Settings window. You can click the **Select All** button, which displays all symbols in that physics interface by selecting all **Show physics symbols** check boxes in the Settings windows for the physics features that include symbols. Similarly, the **Clear All** button clears all **Show physics symbols** check boxes in the individual physics features.



For a more detailed description of Physics Symbols, see [Physics Symbols](#) in the *COMSOL Multiphysics Reference Manual*.

PHYSICAL MODEL


Enter a **Reference pressure level** p_{ref} (SI unit: Pa). The default value is 1 [atm].

GRAVITY EFFECTS

Enter a value for the acceleration of gravity. The default value **g_const** is the predefined standard acceleration of gravity on Earth.

Note that by default there are no gravity effects added to the Layered Darcy's Law interface. To activate the acceleration of gravity, add the **Gravity** feature node from the **Physics** toolbar or right-click **Layered Darcy's Law** to select it from the context menu.

DISCRETIZATION

To display all settings available in this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.

You can choose the order of the shape functions used for the pressure variable solved by the Layered Darcy's Law interface. The default shape functions are **Quadratic Lagrange**.

The **Compute boundary fluxes** check box is not activated by default. When this option is selected, the solver computes variables storing accurate boundary fluxes from each boundary into the adjacent domain.

If the check box is cleared, the COMSOL Multiphysics software instead computes the flux variables from the dependent variables using extrapolation, which is less accurate in postprocessing results, but does not create extra dependent variables on the boundaries for the fluxes.

Also the **Apply smoothing to boundary fluxes** check box is available if the previous check box is selected. The smoothing can provide a better behaved flux value close to singularities.

The **Value type when using splitting of complex variables** setting should in most pure mass transport problems be set to **Real**, which is the default. It makes sure that the dependent variable does not get affected by small imaginary contributions, which can

occur, for example, when combining a Time Dependent or Stationary study with a frequency-domain study.



- For details about the settings for boundary fluxes, see [Computing Accurate Fluxes](#) in the *COMSOL Multiphysics Reference Manual*.
- For more information on splitting complex-valued variables, see [Splitting Complex-Valued Variables](#) in the *COMSOL Multiphysics Reference Manual*.

DEPENDENT VARIABLES

The dependent variable (field variable) is the **Pressure** *p*. The name can be changed but the names of fields and dependent variables must be unique within a model.



- [Boundary, Edge, and Point Nodes for the Layered Darcy’s Law Interface](#)
- [Theory for the Layered Darcy’s Law Interface](#)
- [Physical Constants](#) in the *COMSOL Multiphysics Reference Manual*

Boundary, Edge, and Point Nodes for the Layered Darcy’s Law Interface

The [Layered Darcy’s Law Interface](#) has the following boundary, edge, and point nodes, which 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).

BOUNDARY NODES

- [Fluid and Matrix Properties](#)
- [Gravity](#)
- [Initial Values](#)
- [Storage Model](#)
- [Mass Source](#)
- [Poroelastic Storage](#)
- [Richards' Equation Model](#)

BOUNDARY INTERFACE NODES

- [Pressure, Interface](#)
- [Mass Flux, Interface](#)
- [Inlet, Interface](#)
- [Outlet, Interface](#)
- [Symmetry, Interface](#)
- [No Flow, Interface](#)
- [Flux Discontinuity, Interface](#)
- [Interior Wall, Interface](#)
- [Thin Barrier, Interface](#)

EDGE NODES

The following nodes (listed in alphabetical order) are available on edges. As most of the edge features are very similar to the boundary nodes of the Darcy's Law Interface, those links point to the feature descriptions there. Also take a look at Section [Edge Features](#) below.

- [Continuity](#)
- [Flux Discontinuity](#)
- [Inlet](#)
- [Line Mass Source, Interface](#)
- [Mass Flux](#)
- [No Flow](#)
- [Outlet](#)
- [Pressure](#)
- [Symmetry](#)

POINT NODES

The following nodes (listed in alphabetical order) are available on points:

- [Line Mass Source](#)
- [Point Mass Source](#)



In the *COMSOL Multiphysics Reference Manual*, see [Table 2-4](#) for links to common sections and [Table 2-5](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Fluid and Matrix Properties

The **Fluid and Matrix Properties** node adds the equations for Darcy’s law (excluding any mass sources), and contains settings for the fluid properties and the porous matrix properties such as the effective porosity.

SHELL PROPERTIES

These settings can only be edited if the **Fluid and Matrix Properties** node was added manually. For the default node, the same settings as in the [Shell Properties](#) section of the parent interface are applied.

If you deselect the **Use all layers** check box, you can select individual layers within one layered material. If you have defined more than one layered material in the model, this means that you would have to add another boundary node to define the physics in the remaining material sections.



For a general description of layer selections, see [The Shell Properties and Interface Selection Sections](#) in the *Structural Mechanics Module User’s Guide*.

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 model input is the temperature T and the absolute pressure p_A .

FLUID PROPERTIES

For **Fluid material** you can select **Boundary material** (the default) to use the material defined on the boundary, or another material from the list to use that material's properties for the fluid.

Specify the density and dynamic viscosity as described in the [Fluid](#) section of [The Darcy's Law Interface](#).

MATRIX PROPERTIES

For the porous matrix material, select either **Boundary material** from the **Porous material** list (the default) to use the material defined for the porous domain, or select another material to use that material's properties.

The matrix properties are defined as in [The Darcy's Law Interface](#), so take a look at the [Matrix Properties](#) section for details.

Initial Values

The **Initial Values** node adds an initial value for the pressure that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

SHELL PROPERTIES

The **Shell Properties** settings can only be edited if the **Initial Values** node was added manually. For the default node, the same settings as in the [Shell Properties](#) section of the parent interface are applied.

If you deselect the **Use all layers** check box, you can select individual layers within one layered material. If you need to specify different initial values for different layered materials or for different layers, you have to add one **Initial Values** node for each layered material or for each layer.



For a general description of layer selections, see [The Shell Properties and Interface Selection Sections](#) in the *Structural Mechanics Module User's Guide*.

INITIAL VALUES

Enter a value or expression for the initial value of the **Pressure p** (SI unit: Pa). The default value is 0 Pa.

In case the Subsurface Flow Module is licensed, click the **Pressure head** button to enter a value or expression for H_p (SI unit: m). The default is 0 m. Click the **Hydraulic head** button to enter a value or expression for H (SI unit: m). The default is 0 m.

Richards' Equation Model

The **Richards' Equation Model** node defines the storage and retention models in variably saturated porous media. See the [Unsaturated Porous Medium](#) described for [The Richards' Equation Interface](#).

Note that for the Layered Darcy's Law interface, the **Fluid Properties** and the **Matrix Properties** are defined on the boundary instead of the domain.

SHELL PROPERTIES

In the **Shell Properties** section you determine on which layer the Richard's Equation Model is active. If you deselect the **Use all layers** check box, you can select individual layers within one layered material. To specify different settings for different layered materials or for different layers, you have to add one boundary node for each layered material or for each layer, respectively. See also the [Shell Properties](#) section of the parent interface for more information.



For a general description of layer selections, see [The Shell Properties and Interface Selection Sections](#) in the *Structural Mechanics Module User's Guide*.

Storage Model

The **Storage Model** node adds [Equation 4-5](#) (excluding any mass sources). Use it to define the fluid and porous media properties, including a storage term for the specific storage.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

SHELL PROPERTIES

The settings are the same as for the [Shell Properties](#) section of the [Richards' Equation Model](#) node.

FLUID PROPERTIES

The settings are the same as for the [Fluid and Matrix Properties](#) node except a gas constant type is not required.

MATRIX PROPERTIES

The settings are the same as in the [Matrix Properties](#) section under [Fluid and Matrix Properties](#).

STORAGE MODEL

Select a **Storage** S (SI unit: 1/Pa) — **Linearized storage** (the default) or **User defined**.

- For **User defined** enter a value or expression for **Storage** S (SI unit: 1/Pa).
- For **Linearized storage** it uses the following linear equation to define the storage:

$$S = \epsilon \chi_f + (1 - \epsilon_p) \chi_p$$

This is equivalent to the Reuss average of the fluid and solid compressibilities. Enter the value or expression for each of the following:

- **Compressibility of fluid** χ_f (SI unit: 1/Pa). The default uses values **From material**. For **User defined**, the default is $4 \cdot 10^{-10}$ 1/Pa.

Effective compressibility of matrix χ_p (SI unit: 1/Pa). The default is $1 \cdot 10^{-4}$ 1/Pa).

Poroelastic Storage

The **Poroelastic Storage** node adds [Equation 4-4](#) and [Equation 4-5](#) (excluding any mass sources). Use it to define the fluid and porous media properties, including a storage term to account for the Poroelasticity multiphysics coupling.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

SHELL PROPERTIES

The settings are the same as for the [Shell Properties](#) section of the [Richards' Equation Model](#) node.

FLUID PROPERTIES

The settings are the same as for the [Storage Model](#) node except a gas constant type is not required. There is also an option to define the fluid compressibility.

Compressibility

Select the **Compressibility of fluid** χ_f (SI unit: 1/Pa). The default uses values **From material** as defined by the Fluid material selected. For **User defined** the default is $4 \cdot 10^{-10}$ 1/Pa.

MATRIX PROPERTIES

The settings are the same as in the [Matrix Properties](#) section under [Fluid and Matrix Properties](#).

Gravity

The **Gravity** node adds the acceleration of gravity to the simulation model.

SHELL PROPERTIES

The settings are the same as for the [Shell Properties](#) section of the [Richards' Equation Model](#) node.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes. It can be used when prescribing the direction of the gravitational forces.

GRAVITY

Specify either the acceleration vector or — in case you have a Subsurface Flow Module license — the elevation.

When **Acceleration** is selected from the list, enter the components of the **Gravity vector** \mathbf{g} . The default value is $\mathbf{g_const}$ which is the physical constant having the value 9.8066 m/s^2 .

When **Elevation** is selected from the list, specify the **Elevation** D (SI unit: m). Select the check box **Specify reference position** to define a reference elevation.

Mass Source

The **Mass Source** node adds a mass source Q_m , which appears on the right-hand side of the Darcy's Law equation:

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

SHELL PROPERTIES

The settings are the same as for the [Shell Properties](#) section of the [Richards' Equation Model](#) node.

MASS SOURCE

Enter a value or expression for the **Mass source** Q_m (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$). The default is $0 \text{ kg}/(\text{m}^3 \cdot \text{s})$.

Boundary Interface Features

The Layered Darcy's Law interface has several boundary interface nodes and subnodes available. These nodes apply boundary conditions to the exterior interfaces of a layered shell or to the interfaces between its layers.



Pressure, Interface

Use the **Pressure, Interface** node to specify the pressure on the exterior interfaces of a layered shell or on the interfaces between its layers.

In many cases the distribution of pressure is known, giving a Dirichlet condition $p = p_0$ where p_0 is a known pressure given as a number, a distribution, or an expression involving time, t , for example.

INTERFACE SELECTION

Set the interfaces for which the pressure should be specified. These can be internal interfaces between layers or the top and bottom surfaces.

The available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. 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](#) in the *Heat Transfer Module User's Guide*.

PRESSURE

Enter a value or expression for the **Pressure** p_0 (SI unit: Pa).

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.



For detailed information about the Constraint settings menu see section [Constraint Settings](#) in the *COMSOL Multiphysics Reference Manual*.

Mass Flux, Interface

Use the **Mass Flux, Interface** node to add a mass flux on the exterior interfaces of a layered shell.

It is often possible to determine the mass flux from the pumping rate or from measurements. With this boundary condition, positive values correspond to flow into the model domain:

$$-\mathbf{n} \cdot \rho \mathbf{u} = N_0$$

where N_0 is a value or expression for the specified inward (or outward) Darcy flux.

INTERFACE SELECTION

Set the interfaces for which the mass flux should be specified. Note that it can only be applied to exterior interfaces of the layered shell.

The available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces** (which is the default setting) and **Selected interfaces**. With the last option you can select a set of interfaces and the layered material for which the boundary condition should be active. Note that the check boxes for internal interfaces between layers are deactivated for this node.


The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. 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](#) in the *Heat Transfer Module User's Guide*.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.



For detailed information about the Constraint settings menu see section [Constraint Settings](#) in the *COMSOL Multiphysics Reference Manual*.

MASS FLUX, INTERFACE

Enter a value or expression for the **Inward mass flux** N_0 . A positive value of N_0 represents an inward mass flux whereas a negative value represents an outward mass flux. The units are based on the geometric entity: **Boundaries**: (SI unit: $\text{kg}/(\text{m}^2 \cdot \text{s})$).

Inlet, Interface

The **Inlet, Interface** node adds an inflow condition to the exterior interfaces of a layered shell.

INTERFACE SELECTION

Set the interfaces for which the inflow should be specified. Note that the **Inlet, Interface** node can only be applied to exterior interfaces of the layered shell. The settings are the same as for the [Interface Selection](#) section of the [Mass Flux, Interface](#) node.

BOUNDARY CONDITION

The available **Boundary Condition** options for an inlet are **Velocity**, **Pressure**, and **Mass flow**. After selecting a boundary condition from the list, a section with the same or a similar name displays underneath. For example, if **Velocity** is selected, a **Velocity** section, where further settings are defined, is displayed.

VELOCITY

Enter a value or expression for the **Normal inflow velocity** U_0 (SI unit: m/s). It is specified as $\mathbf{u} = -\mathbf{n}U_0$, where \mathbf{n} is the face normal and U_0 is the normal inflow speed.

A positive value of U_0 represents an inflow velocity. A negative value represents an outflow velocity.

PRESSURE

Enter a value or expression for the inlet **Pressure** p_0 (SI unit: Pa).

MASS FLOW

If you select **Mass flow** as the inlet condition, you have to specify the **Mass flow type**, either as total **Mass flow rate** M_0 (SI unit: kg/s), or the **Pointwise mass flux** N_0 (SI unit: kg/(m²·s)).

With **Mass flow rate** boundary interface condition, positive values correspond to flow into the shell:

$$-\int_{\partial\Omega} \rho(\mathbf{u} \cdot \mathbf{n}) dS = M_0$$


where M_0 is a value or expression for the specified inward (or outward) Darcy flux.

For the **Pointwise mass flux** boundary interface condition, positive values correspond to flow into the shell:

$$-\mathbf{n} \cdot \rho \mathbf{u} = N_0$$

where N_0 is a value or expression for the specified inward (or outward) Darcy flux.

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.



For detailed information about the Constraint settings menu see section [Constraint Settings](#) in the *COMSOL Multiphysics Reference Manual*.

Outlet, Interface

The **Outlet, Interface** node adds an outflow condition to the exterior interfaces of a layered shell.

INTERFACE SELECTION

Set the interfaces for which the outflow should be specified. Note that the **Outlet, Interface** node can only be applied to exterior interfaces of the layered shell. The settings are the same as for the [Interface Selection](#) section of the [Mass Flux, Interface](#) node.

BOUNDARY CONDITION

The available **Boundary Condition** options for an outlet are **Velocity** and **Pressure**. After selecting a boundary condition from the list, a section with the same or a similar name displays underneath. For example, if **Velocity** is selected, a **Velocity** section, where further settings are defined, is displayed.


VELOCITY

Enter a value or expression for the **Normal outflow velocity** U_0 (SI unit: m/s). It is specified as $\mathbf{u} = \mathbf{n}U_0$, so a positive value of U_0 represents an outflow velocity whereas a negative value represents an inflow velocity.

PRESSURE

Similar to the inlet pressure option, the outlet pressure on a boundary interface can be specified. Enter a value or expression for the **Pressure** p_0 (SI unit: Pa).

CONSTRAINT SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box.



For detailed information about the Constraint settings menu see section [Constraint Settings](#) in the *COMSOL Multiphysics Reference Manual*.

Symmetry, Interface



The **Symmetry, Interface** node adds a symmetry boundary condition to the external boundaries of a layered shell. The following condition implements the symmetry condition on an axis or a flow divide:

$$-\mathbf{n} \cdot \rho \mathbf{u} = 0$$

INTERFACE SELECTION

Set the interfaces for which the symmetry condition should be specified. Note that it can only be applied to exterior interfaces of the layered shell.

The available options in the **Apply to** list are **Top interface**, **Bottom interface**, **Exterior interfaces** (which is the default setting), and **Selected interfaces**. With the last option you can select a set of interfaces and the layered material for which the boundary condition should be active. Note that the check boxes for internal interfaces between layers are deactivated for the symmetry, interface node.

The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. 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](#) in the *Heat Transfer Module User's Guide*.

No Flow, Interface

The **No Flow, Interface** states that there is no flow across impermeable external boundaries of a layered shell. The mathematical formulation is:

$$-\mathbf{n} \cdot \rho \mathbf{u} = 0$$

where \mathbf{n} is the vector normal to the boundary.

INTERFACE SELECTION

Set the interfaces for which the no flow condition should be specified. Note that this node can only be applied to exterior interfaces of the layered shell. The settings are the same as for the [Interface Selection](#) section of the [Mass Flux, Interface](#) node.

Flux Discontinuity, Interface

Use the **Flux Discontinuity, Interface** node to specify a mass flux discontinuity through an interior interface of a layered shell. The condition is represented by the following equation:

$$-\mathbf{n} \cdot \rho (\mathbf{u}_1 - \mathbf{u}_2) = N_0$$



In this equation, \mathbf{n} is the vector normal (perpendicular) to the interior layer interface, ρ is the fluid density, \mathbf{u}_1 and \mathbf{u}_2 are the Darcy velocities in the adjacent domains, and N_0 is a specified value or expression for the flux discontinuity.

For this boundary condition, a positive value of N_0 corresponds to a flow discontinuity in the opposite direction to the normal vector of the interior layer interface.

INTERFACE SELECTION

Set the interface for which the flux discontinuity should be specified. Note that this node can only be applied to interior interfaces of a layered shell.

The available options in the **Apply to** list are **Interior interfaces**, which is the default setting, and **Selected interfaces**. With the last option you can select a set of interfaces and the layered material for which the boundary interface condition should be active. Note that the check boxes for exterior interfaces of the layered material are deactivated for this node.

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](#) in the *Heat Transfer Module User's Guide*.

FLUX DISCONTINUITY, INTERFACE

Enter a value or expression for the **Inward mass flux** N_0 (SI unit: $\text{kg}/(\text{m}^2 \cdot \text{s})$). A positive value of N_0 represents a mass flux discontinuity in the opposite direction to the normal vector of the interior boundary.



Interior Wall, Interface

The **Interior Wall, Interface** condition can only be applied on interior interfaces of a layered shell. It is similar to the No Flux, Interface condition, except that it applies on interior interfaces and therefore on both sides of the interfaces. It allows discontinuities of velocity and pressure across the layer interface.

INTERFACE SELECTION

Set the interface for which the **Interior Wall, Interface** condition should be specified. Note that this node can only be applied to interior interfaces of a layered shell.

The available options in the **Apply to** list are **Interior interfaces**, which is the default setting, and **Selected interfaces**. With the last option you can select a set of interfaces and the layered material for which the boundary interface condition should be active. Note that the check boxes for exterior interfaces of the layered material are deactivated for this node.

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](#) in the *Heat Transfer Module User's Guide*.

Thin Barrier, Interface

The **Thin Barrier, Interface** condition models interior permeable interfaces like membranes, permeable walls, geotextiles, or perforated plates as thin permeable barriers. It can only be applied on interior interfaces of layered shells.

At the thin barrier interface, a discontinuity in the velocity field is modeled using the following equations:

$$-\mathbf{n}_u \cdot (\rho \mathbf{u})_u = \frac{\rho(p_d - p_u) \kappa_b}{\mu d_b}$$

$$-\mathbf{n}_d \cdot (\rho \mathbf{u})_d = \frac{\rho(p_u - p_d) \kappa_b}{\mu d_b}$$

where \mathbf{n} is the vector normal to the interface, \mathbf{u} the Darcy velocity, ρ the density and p the pressure. Subscript d denotes the downside of the interface, and u denotes the upside of the interface. κ_b is the permeability of the thin barrier material, d_b the barrier thickness.

INTERFACE SELECTION

Set the interface for which the **Thin Barrier, Interface** node should be specified. Note that it can only be applied to interior interfaces of layered shells.

The settings are the same as for the [Interface Selection](#) section of the [Interior Wall, Interface](#) node.

COORDINATE SYSTEM SELECTION

The **Global coordinate system** is selected by default. The **Coordinate system** list contains any additional coordinate systems that the model includes. It can be used when prescribing the direction of the interface normal and of the velocity.

WALL

Enter a value or expression for the **Permeability** κ_b (SI unit: m^2). The default Permeability κ_b uses the value **From material**. 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.

Enter a value or expression for the **Thickness** d_b (SI unit: m, the default is 0.1 m).

Edge Features

Most of the edge feature nodes available with the Layered Darcy's Law Interface are very similar to the corresponding [Boundary Interface Features](#) nodes and are therefore not further described here. Instead of the Interface selection section the edge feature nodes have a **Shell Properties** section like the boundary nodes. The settings are the same as for example the [Shell Properties](#) section of the [Richards' Equation Model](#) node.

However, there are two special edge feature nodes which have no corresponding boundary interface node and are described below.



Line Mass Source, Interface

The **Line Mass Source, Interface** node adds mass flow originating from a tube of infinitely small radius to edges of layer interfaces of layered shells.

INTERFACE SELECTION

Set the interfaces on whose edges the **Line Mass Source, Interface** condition should be applied. These can be internal interfaces between layers or the top and bottom surfaces.

The available options in the **Apply to** list are **Top interface** (the default setting), **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal

vector. 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](#) in the *Heat Transfer Module User's Guide*.

LINE MASS SOURCE, INTERFACE

Enter a value or expression for the source strength, N_0 (SI unit: kg/(m·s)). A positive value results in mass injection from the line into the computational domain, and a negative value means that the mass is removed from the computational domain.

Line sources located on a boundary affect the adjacent computational domains. This effect makes the physical strength of a line source located in a symmetry plane twice the given strength.



For more information see [Mass Sources for Fluid Flow](#) in the *COMSOL Multiphysics Reference Manual*.


Continuity

Use this node to specify the side-by-side continuity conditions on the pressure field at an edge between two layered materials. By default, the pressure is discontinuous at the interface between two layered materials.

The continuity constraint is added on the common edge in the through-thickness direction.

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. An Offset can be specified on the destination side which decides the actual connection area between the two layered materials.

You can swap the source and destination by clicking **Swap Source and Destination** ()

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**



Point Features

The Layered Darcy's Law Interface has the following special point feature nodes:

Line Mass Source

The **Line Mass Source** node adds mass flow originating from a tube of infinitely small radius. In case of a point feature, the tube lies along the virtual layer edge perpendicular to the layer surface.

SHELL PROPERTIES

In the **Shell Properties** section you determine on which layer edge the Line Mass Source is defined. You can select individual layers within one layered material. To specify different settings for different layered materials or for different layers within one layered material, you have to add one boundary node for each layered material or for each layer, respectively. You can visualize the selected layers by clicking the **Layer Cross Section Preview** () and **Layer 3D Preview** () buttons.

LINE MASS SOURCE

Enter a value or expression for the source strength, N_0 (SI unit: kg/(m·s)). A positive value results in mass injection from the line into the computational domain, and a negative value means that the mass is removed from the computational domain.





For more information see [Mass Sources for Fluid Flow](#) in the *COMSOL Multiphysics Reference Manual*.

Point Mass Source

The **Point Mass Source** node models mass flow originating from an infinitely small sphere centered around a point which lies on a layer interface of a layered material.

INTERFACE SELECTION

Set the interfaces on whose points the **Point Mass Source** condition should be applied. These can be internal interfaces between layers or the top and bottom surfaces.

The available options in the **Apply to** list are **Top interface** (the default setting), **Bottom interface**, **Exterior interfaces**, **Interior interfaces**, **All interfaces**, and **Selected interfaces**. With the last option you can select any set of interfaces for a given layered material, by clearing the check boxes corresponding to layer interfaces where the node should not be applied in the **Selection** table. The top and bottom interfaces refer respectively to the upside and downside of the boundary, defined from the orientation of the normal vector. 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](#) in the *Heat Transfer Module User's Guide*.

POINT MASS SOURCE

Enter a value or expression for the source strength, N_0 (SI unit: kg/(s)). A positive value results in mass injection from the point into the computational domain, and a negative value means that the mass is removed from the computational domain.

Theory for the Brinkman Equations Interface

The [Brinkman Equations Interface](#) theory is described in this section:

- [About the Brinkman Equations](#)
- [Brinkman Equations Theory](#)
- [References for the Brinkman Equations Interface](#)

About the Brinkman Equations

The *Brinkman equations* describe fast-moving fluids in porous media with the kinetic potential from fluid velocity, pressure, and gravity to drive the flow. These equations extend Darcy's law to describe the dissipation of the kinetic energy by viscous shear as with the Navier-Stokes equations. Consequently, this physics interface well suits transitions between slow flow in porous media governed by Darcy's law and fast flow in channels described by the Navier-Stokes equations. Interesting uses of a Brinkman and Navier-Stokes coupling include modeling of the hyporheic zone near a river, the flow of oil through a reservoir to a well perforation, and non-Newtonian flows.

In porous domains, the flow variables and fluid properties are defined at any point inside the medium by means of averaging of the actual variables and properties over a certain volume surrounding the point. This control volume must be small compared to the typical macroscopic dimensions of the problem, but it must be large enough to contain many pores and solid matrix elements.

Porosity is defined as the fraction of the control volume that is occupied by pores. Thus, the porosity can vary from zero for pure solid regions to unity for domains of free flow.

The physical properties of the fluid, such as density and viscosity, are defined as *intrinsic volume averages* that correspond to a unit volume of the pores. Defined this way, they present the relevant physical parameters that can be measured experimentally, and they are assumed to be continuous with the corresponding parameters in the adjacent free flow.

The flow velocity is defined as a *superficial volume average*, and it corresponds to a unit volume of the medium including both the pores and the matrix. It is sometimes

called the Darcy velocity, defined as the volume flow rate per unit cross section of the medium. Such a definition makes the velocity field continuous across the boundaries between porous regions and regions of free flow.

Brinkman Equations Theory

The dependent variables in the Brinkman equations are the Darcy velocity and the pressure. The flow in porous media is governed by a combination of the continuity equation and the momentum equation, which together form the Brinkman equations:

$$\frac{\partial}{\partial t}(\epsilon_p \rho) + \nabla \cdot (\rho \mathbf{u}) = Q_m \quad (4-25)$$

$$\begin{aligned} \frac{\rho}{\epsilon_p} \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \frac{\mathbf{u}}{\epsilon_p} \right) = \\ -\nabla p + \nabla \cdot \left[\frac{1}{\epsilon_p} \left\{ \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right\} \right] - \left(\kappa^{-1} \mu + \frac{Q_m}{\epsilon_p^2} \right) \mathbf{u} + \mathbf{F} \end{aligned} \quad (4-26)$$

In these equations:

- μ (SI unit: kg/(m·s)) is the dynamic viscosity of the fluid
- \mathbf{u} (SI unit: m/s) is the velocity vector
- ρ (SI unit: kg/m³) is the density of the fluid
- p (SI unit: Pa) is the pressure
- ϵ_p is the porosity
- κ (SI unit: m²) is the permeability of the porous medium, and
- Q_m (SI unit: kg/(m³·s)) is a mass source or sink

Influence of gravity and other volume forces can be accounted for via the force term \mathbf{F} (SI unit: kg/(m²·s²)).

When the **Neglect inertial term (Stokes flow)** check box is selected, the term $(\mathbf{u} \cdot \nabla)(\mathbf{u}/\epsilon_p)$ on the left-hand side of Equation 4-26 is disabled.

The mass source, Q_m , accounts for mass deposit and mass creation within the domains. The mass exchange is assumed to occur at zero velocity.

The Forchheimer and Ergun drag options add a viscous force proportional to the square of the fluid velocity, $\mathbf{F} = -\rho \beta |\mathbf{u}| \mathbf{u}$, on the right-hand side of Equation 4-26; see [References for the Darcy's Law Interface](#) for details.

In case of a flow with variable density, Equation 4-25 and Equation 4-26 must be solved together with the equation of state that relates the density to the temperature and pressure (for instance the ideal gas law).

For incompressible flow, the density stays constant in any fluid particle, which can be expressed as

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



and the continuity equation (Equation 4-25) reduces to

$$\rho \nabla \cdot \mathbf{u} = Q_m$$

References for the Brinkman Equations Interface

1. D. Nield and A. Bejan, *Convection in Porous Media*, 3rd ed., Springer, 2006.
2. M. Le Bars and M.G. Worster, “Interfacial Conditions Between a Pure Fluid and a Porous Medium: Implications for Binary Alloy Solidification,” *J. Fluid Mech.*, vol. 550, pp. 149–173, 2006.

The Brinkman Equations Interface


The **Brinkman Equations (br)** interface (), found under the **Porous Media and Subsurface Flow** branch () when adding a physics interface, is used to compute fluid velocity and pressure fields of single-phase flow in porous media in the laminar flow regime. The physics interface extends Darcy's law to describe the dissipation of the kinetic energy by viscous shear, similar to the Navier–Stokes equations. Fluids with varying density can be included at Mach numbers below 0.3. Also the viscosity of a fluid can vary, for example, to describe non-Newtonian fluids. To simplify the equations, select the Stokes–Brinkman flow feature to reduce the dependence on inertial effects when the Reynolds number is significantly less than 1. The physics interface can be used for stationary and time-dependent analyses.

The main node is the Porous Medium feature, which adds the Brinkman equations and provides interfaces for defining the fluid material and the porous matrix.

When this physics interface is added, the following default nodes are also added in the **Model Builder — Porous Medium, Wall** (the default boundary type, using **No slip** as the default boundary condition), and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Brinkman Equations** to select physics features from the context menu.

The boundary conditions are essentially the same as for the Laminar Flow interface. Differences exist for the following boundary types: Outlet, Symmetry, Open Boundary, and Boundary Stress where the viscous part of the stress is divided by the porosity to appear as

$$\frac{1}{\varepsilon_p} \left\{ \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right\}$$

In the *COMSOL Multiphysics Reference Manual* see [Table 2-4](#) for links to common sections such as **Discretization**, **Consistent Stabilization**, and **Inconsistent Stabilization**, and **Advanced Settings** sections, all accessed by clicking the **Show** button () and choosing the applicable option. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

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 `br`.

PHYSICAL MODEL

This node specifies the properties of the Brinkman Equations interface, which describe the overall type of fluid flow model.

Compressibility

By default the physics interface uses the **Incompressible flow** formulation of the Brinkman equations to model constant density flow. Alternatively, from the **Compressibility** list select **Weakly compressible flow** which should be used when the pressure dependency of the density can be neglected, or **Compressible flow (Ma<0.3)** to solve for the full compressible flow of which, however, the Mach number must be below 0.3.

Swirl Flow

For 2D axisymmetric models, select the **Swirl flow** check box to include the swirl velocity component, that is the velocity component u_φ in the azimuthal direction. While u_φ can be nonzero, there can be no gradients in the φ direction.

Note that this feature is only available for specific modules. See <https://www.comsol.com/products/specifications/> for a detailed overview.

Neglect Inertial Term (Stokes–Brinkman Flow)

The **Neglect inertial term (Stokes–Brinkman)** check box is selected by default to model flow at low Reynolds numbers for which the inertial term can be neglected. This results in the linear Stokes–Brinkman equations.

Include Gravity

Gravity is not included by default. Select the **Include gravity** check box to activate the acceleration of gravity. This automatically adds a global **Gravity** feature node to the interface model tree, and the buoyancy force is included in the Equations.

Also, when the **Include gravity** check box is selected, the **Use reduced pressure** option changes the pressure formulation from using the total pressure (default) to using the reduced pressure. This option is suitable for configurations where the density changes

are very small; otherwise, the default formulation can be used. For more information, see [Gravity](#).

Porous Treatment of No Slip Condition

Choose how the **No Slip** condition on **Wall** boundaries and **Interior Wall** boundaries adjacent to **Porous Medium** should be treated. The options are **Standard no slip formulation** (default) and **Porous slip**. The latter option provides a unified treatment when the porous matrix is fully resolved as well as when it is under resolved ensuring a smooth transition between regions with different resolutions; see No Slip under [Wall](#) in the [The Laminar Flow and Creeping Flow Interfaces](#) section.

Reference Pressure Level

Enter a **Reference pressure level** p_{ref} (SI unit: Pa). The default value is 1 [atm].

Reference Temperature Level

Enter a **Reference temperature level** T_{ref} (SI unit: K). The default value is 293.15 [K].

Reference Position

If **Include gravity** is selected, a **Reference position** r_{ref} (SI unit: m) can be specified which is then used for the calculation of the hydrostatic pressure.


TURBULENCE

Turbulent flow can be simulated by changing the **Turbulence model type** to **RANS** (Reynolds-Averaged Navier–Stokes). If turbulent flow is activated, one of the following **Turbulence models** can be chosen: **Algebraic yPlus**, **L-VEL**, and **k-ε**, and **Wall functions** are preset for **Wall treatment**. For more information about turbulence modeling, see [Theory for the Turbulent Flow Interfaces](#) in the *CFD Module User's Guide*.

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface — the **Velocity field** \mathbf{u} (SI unit: m/s) and its components, and the **Pressure** p (SI unit: Pa).

ADVANCED SETTINGS

To display this section, click the **Show More Options** button () and select **Advanced Physics Options** in the **Show More Options** dialog box. Normally these settings do not need to be changed.

The **Use pseudo time stepping for stationary equation form** option adds pseudo time derivatives to the equation when the **Stationary equation** form is used in order to speed up convergence. When selected, a **CFL number expression** should also be defined. For

the default **Automatic** option, the local CFL number (from the Courant–Friedrichs–Lewy condition) is determined by a PID regulator.



- [Domain, Boundary, Point, and Pair Nodes for the Brinkman Equations Interface](#)
- [Theory for the Brinkman Equations Interface](#)
- [Pseudo Time Stepping for Laminar Flow Models](#) and [Pseudo Time Stepping](#) in the *COMSOL Multiphysics Reference Manual*
- [Numerical Stability — Stabilization Techniques for Fluid Flow](#)
- [Discontinuous Galerkin Formulation](#)

Domain, Boundary, Point, and Pair Nodes for the Brinkman Equations Interface

The [Brinkman Equations Interface](#) has the following domain, boundary, point, and pair nodes, listed in alphabetical order, 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.

These nodes are described in this section:

- [Porous Medium](#)
- [Initial Values](#)
- [Mass Source](#)
- [Volume Force](#)
- [Fluid Properties](#)

The following nodes (listed in alphabetical order) are described for the Laminar Flow interface :

- [Flow Continuity](#)
- [Inlet](#)
- [Line Mass Source](#)
- [Boundary Stress](#)
- [Outlet](#)
- [Open Boundary](#)
- [Periodic Flow Condition](#)
- [Point Mass Source](#)
- [Pressure Point Constraint](#)
- [Symmetry](#)
- [Wall](#)



In the *COMSOL Multiphysics Reference Manual* see [Table 2-4](#) for links to common sections and [Table 2-5](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Porous Medium

The **Porous Medium** node adds the Brinkman equations: [Equation 4-25](#) and [Equation 4-26](#) (excluding any mass sources). It has two default subfeatures — the [Fluid](#) subfeature where the fluid density ρ (SI unit: kg/m^3) and dynamic viscosity μ (SI unit: $\text{Pa}\cdot\text{s}$) are defined and the [Porous Matrix](#) subfeature where the permeability κ (SI unit: m^2) and porosity ε_p (dimensionless) are specified.

FLOW MODEL

Define if the flow follows a linear or nonlinear pressure-velocity relationship.

- **Darcian flow** (default) defines a linear relationship.
- **Non-Darcian flow** defines a nonlinear relationship due to turbulence or inertial effects.

Fluid

This node defines the material properties of the fluid of the [Porous Medium](#) parent node.

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.

FLUID PROPERTIES

Specify the **Density** ρ and the **Dynamic viscosity** μ (SI unit: Pa·s) of the fluid. The defaults use values **From material**, for **User defined** enter a value or expression.

The dynamic viscosity describes the relationship between the shear stresses and the shear rate in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick, such as oil, have a higher viscosity.

Porous Matrix

This node defines the porosity and material properties of the solid matrix of the **Porous Medium** parent node.

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.

MATRIX PROPERTIES

The **Porosity** ε_p (a dimensionless number between 0 and 1) uses by default the value **From material**. For **User defined** the default value is 1.

Select a **Permeability model** to specify the capacity of the porous material to transmit flow. Depending on the **Flow model** selection in the parent **Porous Medium** feature node you can choose between the following options:

- **Permeability** (the default) to directly enter the permeability of the porous matrix or **Kozeny-Carman** to define the permeability from the porosity and mean **Particle diameter** for **Darcian** flow or
- **Forchheimer** or **Ergun** for **Non-Darcian** flow.

Note that some of these options are only available for certain modules. For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>.

The default **Permeability** κ (SI unit: m^2) uses the value **From material**, as defined by the **Porous material** list. 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.

For **Kozeny-Carman**, enter the mean **Particle diameter** d_p (SI unit: m), the default value is 0.5 mm.

In case a **Non-Darcian** flow model is chosen in the parent node, **Forchheimer** is the default selection for the **Permeability model**. For **Forchheimer**, the default **Permeability** κ (SI unit: m^2) uses the value **From material**. Furthermore, the dimensionless **Forchheimer parameter** c_F can be defined. The default value is 0.55.

If **Ergun** is selected, enter the mean **Particle diameter** d_p (default value: 0.5 mm). The permeability κ is then calculated using [Equation 2-4](#).



- [About Darcian and Non-Darcian Flow](#)
- [Permeability Models](#)

Mass Source

The **Mass Source** node adds a mass source (or mass sink) Q_m to the right-hand side of the continuity equation: [Equation 4-25](#). This term accounts for mass deposit and/or mass creation in porous domains. The physics interface assumes that the mass exchange occurs at zero velocity.

$$\frac{\partial}{\partial t}(\epsilon_p \rho) + \nabla \cdot (\rho \mathbf{u}) = Q_m \quad (4-27)$$

DOMAIN SELECTION

Only Porous Matrix domains are available.

MASS SOURCE

Enter a value or expression for the **Source term** Q_{br} (SI unit: $\text{kg}/(\text{m}^3 \cdot \text{s})$). The default is $0 \text{ kg}/(\text{m}^3 \cdot \text{s})$.

Volume Force

Use the **Volume Force** node to specify the force \mathbf{F} on the right-hand side of [Equation 4-26](#). It then acts on each fluid element in the specified domains.

$$\frac{\rho}{\epsilon_p} \left(\frac{\partial \mathbf{u}}{\partial t} + (\mathbf{u} \cdot \nabla) \frac{\mathbf{u}}{\epsilon_p} \right) = -\nabla p + \nabla \cdot \left[\frac{1}{\epsilon_p} \left\{ \mu (\nabla \mathbf{u} + (\nabla \mathbf{u})^T) - \frac{2}{3} \mu (\nabla \cdot \mathbf{u}) \mathbf{I} \right\} \right] - \left(\kappa^{-1} \mu + \frac{Q_m}{\epsilon_p^2} \right) \mathbf{u} + \mathbf{F} \quad (4-28)$$

VOLUME FORCE

Enter the components of **Volume force** \mathbf{F} (SI unit: N/m³).

Initial Values

The **Initial Values** node adds initial values for the velocity field and the pressure that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter initial values or expressions for the **Velocity field** \mathbf{u} (SI unit: m/s) and the **Pressure** p (SI unit: Pa). The default values are 0 m/s and 0 Pa, respectively.

Fluid Properties

The **Fluid Properties** node adds the momentum and continuity equations to solve for free flow in nonporous domains. The node also provides an interface for defining the material properties of the fluid.

MODEL INPUTS

Fluid properties, such as density and viscosity, can be defined through user inputs, variables or by selecting a material. For the latter option, additional inputs — for example, temperature or pressure — may be required to define these properties.

Temperature


By default, the single-phase flow interfaces are set to model isothermal flow. Hence, the **Temperature** is **User defined** and defaults to 293.15 K. If a Heat Transfer interface is included in the component, the temperature may alternatively be selected from this physics interface. All physics interfaces have their own tags (**Name**). For example, if a Heat Transfer in Fluids interface is included in the component, the **Temperature (ht)** option is available.

Absolute Pressure

This input appears when a material requires the absolute pressure as a model input. The absolute pressure is used to evaluate material properties, but it also relates to the value of the calculated pressure field. There are generally two ways to calculate the pressure when describing fluid flow: either to solve for the absolute pressure or for a pressure (often denoted gauge pressure) that relates to the absolute pressure through a reference pressure.

The choice of pressure variable depends on the system of equations being solved. For example, in a unidirectional incompressible flow problem, the pressure drop over the modeled domain is probably many orders of magnitude smaller than the atmospheric pressure, which, when included, may reduce the stability and convergence properties of the solver. In other cases, such as when the pressure is part of an expression for the gas volume or the diffusion coefficients, it may be more convenient to solve for the absolute pressure.

The default **Absolute pressure** p_A is $p + p_{\text{ref}}$ where p is the dependent pressure variable from the Navier–Stokes equations, and p_{ref} is from the user input defined at the physics interface level. When p_{ref} is nonzero, the physics interface solves for a gauge pressure. If the pressure field instead is an absolute pressure field, p_{ref} should be set to 0.

The **Absolute pressure** field can be edited by clicking **Make All Model Inputs Editable** () and entering the desired value in the input field.

FLUID PROPERTIES

If density variations with respect to pressure are to be included in the computations, the flow must be set to compressible.

The **Dynamic viscosity** μ describes the relationship between the shear rate and the shear stresses in a fluid. Intuitively, water and air have low viscosities, and substances often described as thick (such as oil) have higher viscosities.

Gravity

The **Gravity** global feature is automatically added when **Include gravity** is selected at the interface level in the **Physical Model** settings. It defines the gravity forces from the **Acceleration of gravity** value. When a turbulence model that solves for the turbulent kinetic energy, k , is used, the option **Include buoyancy-induced turbulence** is available. When selected, the **Buoyancy contribution** is by default set to **Automatic from multiphysics**. Contributions are only obtained from multiphysics couplings that

support buoyancy-induced turbulence, such as Nonisothermal Flow. If the **Buoyancy contribution** is switched to **User defined**, a text field for the **Turbulent Schmidt number** appears. See also [Theory for Buoyancy-Induced Turbulence](#) in the *CFD Module User's Guide*.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

ACCELERATION OF GRAVITY

The **Acceleration of gravity** (SI unit m/s, default value $-g_{\text{const}}\mathbf{e}_z$ in 2D axial symmetry and 3D and $-g_{\text{const}}\mathbf{e}_y$ in 2D) is used to define the gravity forces. It should be a global quantity.

Theory for the Free and Porous Media Flow Interface

The [Free and Porous Media Flow Interface](#) uses the Navier–Stokes equations to describe the flow in open regions, and the Brinkman equations to describe the flow in porous regions.



The same fields, \mathbf{u} and p , are solved for in both the free flow domains and in the porous domains. This means that the pressure in the free fluid and the pressure in the pores is continuous over the interface between a free flow domain and a porous domain. It also means that continuity is enforced between the fluid velocity in the free flow and the Darcy velocity in the porous domain. This treatment is one of several possible models for the physics at the interface. Examples of other models can be found in [Ref. 1](#).

The continuity in \mathbf{u} and p implies a stress discontinuity at the interface between a free-flow domain and a porous domain. The difference corresponds to the stress absorbed by the rigid porous matrix, which is a consequence implicit in the formulations of the Navier–Stokes and Brinkman equations.

Reference for the Free and Porous Media Flow Interface

1. M.L. Bars and M.G. Worster, “Interfacial Conditions Between a Pure Fluid and a Porous Medium: Implications for Binary Alloy Solidification,” *J. Fluid Mech.*, vol. 550, pp. 149–173, 2006.

The Free and Porous Media Flow Interface

The **Free and Porous Media Flow (fp)** interface () is found under the **Porous Media and Subsurface Flow** branch () when adding a physics interface. It is used to compute fluid velocity and pressure fields of single-phase flow where free flow is connected to porous media. The Free and Porous Media Flow interface is used over at least two different domains: a free channel and a porous medium. The physics interface is well suited for transitions between slow flow in porous media, governed by the Brinkman equations, and fast flow in channels described by the Navier–Stokes equations. Fluids with varying density can be included at Mach numbers below 0.3. Also the viscosity of a fluid can vary, for example, to describe non-Newtonian fluids. The physics interface can be used for stationary and time-dependent analyses.

When this physics interface is added, the following default nodes are also added in the **Model Builder — Fluid Properties, Wall, and Initial Values**. Then, from the **Physics** toolbar, add a **Porous Medium** node to be used on the domain selection corresponding to the porous media, or add other nodes that implement, for example, boundary conditions and volume forces. You can also right-click **Free and Porous Media Flow** to select physics features from the context menu.

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 `fp`.

PHYSICAL MODEL

Compressibility

By default the physics interface uses the **Incompressible flow** formulation of the Navier–Stokes and Brinkman equations to model constant density flow. If required, select **Weakly compressible flow** from the **Compressibility** list, to account for small variations in

the density, which are only dependent on the temperature (nonisothermal flow) or **Compressible flow (Ma<0.3)** for fully compressible flow. However, for the flow modeled with this physics interface, the Mach number must be below 0.3.

Swirl Flow

For 2D axisymmetric components, select the **Swirl flow** check box to include the swirl velocity component, that is the velocity component u_ϕ in the azimuthal direction. While u_ϕ can be nonzero, there can be no gradients in the ϕ direction. Note that this feature is only available for specific modules. Visit <https://www.comsol.com/products/specifications/> for a detailed overview.

Neglect Inertial Term

Select the **Neglect inertial term (Stokes flow)** check box if the inertial forces are small compared to the viscous forces.

Enable porous media domains

The **Enable porous media domains** check box is selected by default to solve the Brinkman equations in porous domains. If it is unchecked, the **Porous Medium** node is no longer available in this interface.

Include Gravity

Gravity is not included by default. Select the **Include gravity** check box to activate the acceleration of gravity. This automatically adds a global **Gravity** feature node to the interface model tree, and the buoyancy force is included in the Equations.

Also, when the **Include gravity** check box is selected, the **Use reduced pressure** option changes the pressure formulation from using the total pressure (default) to using the reduced pressure. This option is suitable for configurations where the density changes are very small; otherwise, the default formulation can be used. For more information, see [Gravity](#).

Porous Treatment of No Slip Condition

Choose how the **No Slip** condition on internal boundaries should be treated. The options are **Standard no slip formulation** (default) and **Porous slip**. The latter option provides a unified treatment when the porous matrix is fully resolved as well as when it is under resolved ensuring a smooth transition between regions with different resolutions; see No Slip under **Wall** in the [The Laminar Flow and Creeping Flow Interfaces](#) section.

Reference Pressure Level

Enter a **Reference pressure level** p_{ref} (SI unit: Pa). The default value is 1 [atm].

Reference Temperature

Enter a **Reference temperature** T_{ref} (SI unit: K). The default value is 293.15[K].

Reference Position

If **Include gravity** is selected, a **Reference position** r_{ref} (SI unit: m) can be specified which is then used for the calculation of the hydrostatic pressure.

TURBULENCE

Turbulent flow can be simulated by changing the **Turbulence model type** to **RANS** (Reynolds-Averaged Navier–Stokes) or **Large Eddy Simulation** (which is only available in 3D). If turbulent flow is activated, you can choose from different **Turbulence models** and options for **Wall treatment**. For a description of the different turbulence models, wall treatment options, and turbulence model parameters see [Theory for the Turbulent Flow Interfaces](#) in the *CFD Module User’s Guide*.

DEPENDENT VARIABLES

The following dependent variables (fields) are defined for this physics interface — the **Velocity field** \mathbf{u} (SI unit: m/s) and its components, and the **Pressure** p (SI unit: Pa).



- [Domain, Boundary, Point, and Pair Nodes for the Free and Porous Media Flow Interface](#)
- [Theory for the Free and Porous Media Flow Interface](#)

Domain, Boundary, Point, and Pair Nodes for the Free and Porous Media Flow Interface

[The Free and Porous Media Flow Interface](#) has the following domain, boundary, point, and pair nodes, listed in alphabetical order, 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.

- Fluid Properties
- Gravity
- Initial Values
- Porous Medium
- Mass Source
- Volume Force
- Wall

Note that some features are only available with certain COMSOL products (see <https://www.comsol.com/products/specifications/>).

The following nodes (listed in alphabetical order) are described for the Laminar Flow interface:

- No Viscous Stress
- Flow Continuity
- Inlet
- Line Mass Source
- Outlet
- Open Boundary
- Periodic Flow Condition
- Point Mass Source
- Pressure Point Constraint
- Symmetry



In the *COMSOL Multiphysics Reference Manual* see [Table 2-4](#) for links to common sections and [Table 2-5](#) to common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Fluid Properties

Use the **Fluid Properties** node to define the fluid material, density, and dynamic viscosity.

FLUID PROPERTIES

The default **Fluid material** uses the **Domain material** (the material defined for the domain). Select another material as needed.

The default **Density** ρ (SI unit: kg/m^3) uses values **From material** based on the **Fluid material** selection. For **User defined** enter another value or expression. The default is $0 \text{ kg}/\text{m}^3$.

The **Dynamic viscosity** μ (SI unit: Pa·s) uses values **From material** based on the **Fluid material** selection. For **User defined** enter another value or expression. The default is 0 Pa·s.

Porous Medium

The **Porous Medium** node adds the Brinkman equations (excluding any mass sources) to the interface. It has two default subfeatures — the **Fluid** subfeature where the fluid density ρ (SI unit: kg/m³) and dynamic viscosity μ (SI unit: Pa·s) are defined and the **Porous Matrix** subfeature where the permeability κ (SI unit: m²) and porosity ϵ_p (dimensionless) are specified.

FLOW MODEL

Define if the flow follows a linear or nonlinear pressure-velocity relationship.

- **Darcian flow** (default) defines a linear relationship.
- **Non-Darcian flow** defines a nonlinear relationship due to turbulence or inertial effects.

Fluid

This node defines the material properties of the fluid of the **Porous Medium** parent node.

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.

FLUID PROPERTIES

Specify the **Density** ρ and the **Dynamic viscosity** μ (SI unit: Pa·s) of the fluid. The defaults use values **From material**, for **User defined** enter a value or expression.

The dynamic viscosity describes the relationship between the shear stresses and the shear rate in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick, such as oil, have a higher viscosity.

This node defines the porosity and material properties of the solid matrix of the **Porous Medium** parent node.

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.

MATRIX PROPERTIES

The **Porosity** ε_p (a dimensionless number between 0 and 1) uses by default the value **From material**. For **User defined** the default value is 1 which is equivalent to free flow.

Select a **Permeability model** to specify the capacity of the porous material to transmit flow. Depending on the **Flow model** selection in the parent **Porous Medium** feature node you can choose between the following options:

- **Permeability** (the default) to directly enter the permeability of the porous matrix or **Kozeny-Carman** to define the permeability from the porosity and mean **Particle diameter** for **Darcian** flow or
- **Forchheimer** or **Ergun** for **Non-Darcian** flow.

Note that some of these options are only available for certain modules. For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>.

The default **Permeability** κ (SI unit: m^2) uses the value **From material**, as defined by the **Porous material** list. 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.

For **Kozeny-Carman**, enter the mean **Particle diameter** d_p (SI unit: m), the default value is 0.5 mm.

In case a **Non-Darcian** flow model is chosen in the parent node, **Forchheimer** is the default selection for the **Permeability model**. For **Forchheimer**, the default **Permeability** κ (SI unit: m^2) uses the value **From material**. Furthermore, the dimensionless **Forchheimer parameter** c_F can be defined. The default value is 0.55.

If **Ergun** is selected, enter the mean **Particle diameter** d_p (default value: 0.5 mm). The permeability κ is then calculated using [Equation 2-4](#).



- [About Darcian and Non-Darcian Flow](#)
- [Permeability Models](#)

Mass Source

Enter a value or expression for an optional mass source (or sink) **Source term** Q_m (SI unit: $\text{kg}/(\text{m}^3\cdot\text{s})$). This term accounts for mass deposit or mass creation within porous domains. The physics interface assumes that the mass exchange occurs at zero velocity.

Volume Force

The **VOLUME FORCE** node specifies the force \mathbf{F} on the right-hand side of the Navier–Stokes or Brinkman equations, depending on whether a [Fluid Properties](#) or [Porous Medium](#) node is active on the domain. It then acts on each fluid element in the specified domains.

VOLUME FORCE

Enter the components of the **Volume force** \mathbf{F} (SI unit: N/m^3).

Initial Values

The **Initial Values** node adds initial values for the velocity field and the pressure that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter initial values or expressions for the **Velocity field** \mathbf{u} (SI unit: m/s) and for the **Pressure** p (SI unit: Pa). The default values are 0 m/s and 0 Pa, respectively.

Gravity

The **Gravity** global feature is automatically added when **Include gravity** is selected at the interface level in the **Physical Model** settings. It defines the gravity forces from the **Acceleration of gravity** value. When a turbulence model that solves for the turbulent kinetic energy, k , is used, the option **Include buoyancy-induced turbulence** is available.

When selected, the **Buoyancy contribution** is by default set to **Automatic from multiphysics**. Contributions are only obtained from multiphysics couplings that support buoyancy-induced turbulence, such as Nonisothermal Flow. If the **Buoyancy contribution** is switched to **User defined**, a text field for the **Turbulent Schmidt number** appears. See also [Theory for Buoyancy-Induced Turbulence](#) in the *CFD Module User's Guide*.

This feature requires a specific license. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

ACCELERATION OF GRAVITY

The **Acceleration of gravity** (SI unit m/s, default value $-g_{\text{const}}\mathbf{e}_z$ in 2D axial symmetry and 3D and $-g_{\text{const}}\mathbf{e}_y$ in 2D) is used to define the gravity forces. It should be a global quantity.

Wall

The **Wall** node includes a set of boundary conditions describing fluid-flow conditions at stationary, moving, and leaking walls.

BOUNDARY CONDITION

Select a **Boundary condition** for the wall.

- [No Slip](#)¹
- [Slip](#)
- [Leaking Wall](#)

No Slip

No slip is the default boundary condition for a stationary solid wall for laminar flow (and SST, Low Re k- ϵ , Algebraic yPlus, L-VEL, and Spalart-Allmaras turbulence models). The condition prescribes $\mathbf{u} = 0$; that is, the fluid at the wall is not moving.


Slip

The **Slip** option prescribes a no-penetration condition, $\mathbf{u} \cdot \mathbf{n} = 0$. It is implicitly assumed that there are no viscous effects at the slip wall and hence, no boundary layer develops. From a modeling point of view, this can be a reasonable approximation if the main effect of the wall is to prevent fluid from leaving the domain.

Leaking Wall

This boundary condition may be used to simulate a wall where fluid is leaking into or leaving the domain with the velocity $\mathbf{u} = \mathbf{u}_1$ through a perforated wall. The components of the **Fluid velocity** \mathbf{u}_1 on the leaking wall should be specified.


CONSTRAINT SETTINGS

This section is displayed by clicking the **Show** button () and selecting **Advanced Physics Options**.



- [Theory for the Wall Boundary Condition](#)
- [The Moving Mesh Interface](#) in the *COMSOL Multiphysics Reference Manual*

Chemical Species Transport Interfaces



This chapter describes the physics interfaces found under the **Chemical Species Transport** branch (). It summarizes the physics interfaces including the different fields, material properties, and boundary conditions associated with them, and suggests how and when to apply the conditions.

In this chapter:

- [The Transport of Diluted Species Interface](#)
- [The Transport of Diluted Species in Porous Media Interface](#)
- [The Transport of Diluted Species in Fractures Interface](#)
- [The Moisture Transport Interfaces](#)
- [Theory for the Transport of Diluted Species Interface](#)

[The Moisture Flow Interface](#) as well as [The Reacting Flow in Porous Media Interface](#) are also available under the **Chemical Species Transport** branch. They are described in Chapter [Multiphysics Interfaces and Couplings](#).

The Transport of Diluted Species Interface

The **Transport of Diluted Species (tds)** interface () , found under the **Chemical Species Transport** branch () , is used to calculate the concentration field of a dilute solute in a solvent. Transport and reactions of the species dissolved in a gas, liquid, or solid can be handled with this interface. The driving forces for transport can be diffusion by Fick's law, convection when coupled to a flow field, and migration, when coupled to an electric field.

The interface supports simulation of transport by convection and diffusion in 1D, 2D, and 3D as well as for axisymmetric components in 1D and 2D. The dependent variable is the molar concentration, c . Modeling multiple species transport is possible, whereby the physics interface solves for the molar concentration, c_i , of each species i .

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 **tds**.

DOMAIN SELECTION

If any part of the model geometry should not partake in the mass transfer model, remove that part from the selection list.

OUT-OF-PLANE THICKNESS

For 2D components, the **Thickness** field (default value: 1 m) defines a parameter for the thickness of the geometry perpendicular to the two-dimensional cross-section. Both constant and varying thicknesses are supported. The value of this parameter is used, among other things, to automatically calculate molar flow rates from the total molar flux.

CROSS-SECTIONAL AREA

For 1D components, enter a **Cross-sectional area** A_c (SI unit: m^2) to define a parameter for the area of the geometry perpendicular to the 1D component. Both constant and varying areas are supported. The value of this parameter is used, among other things, to automatically calculate molar flow rates from the total molar flux. The default is 1 m^2 .

TRANSPORT MECHANISMS

Mass transport due to diffusion is always included. Use the check boxes available under **Additional transport mechanisms** to control other transport mechanisms.

Note: Some of the additional transport mechanisms listed below are only available in certain products. For details see <https://www.comsol.com/products/specifications/>.

- By default, the **Convection** check box is selected. Clear the check box to disable convective transport.
- Select the **Migration in electric field** check box to activate transport of ionic species in an electric field. See further the theory section [Adding Transport Through Migration](#).



Transport of ionic species in an electric field is available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

Mass Transport in Porous Media

The **Mass transport in porous media** check box activates functionality specific to species transport in porous media:

- [Porous Medium](#)
- [Unsaturated Porous Medium](#)
- [Porous Electrode Coupling](#)

- Volatilization
- Species Source




Note: Mass transport in porous media is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

CONSISTENT STABILIZATION


To display this sections, click the **Show** button () and select **Stabilization**.

- When the **Crosswind diffusion** check box is selected, a weak term that reduces spurious oscillations is added to the transport equation. The resulting equation system is always nonlinear. There are two options for the **Crosswind diffusion type**:
 - **Do Carmo and Galeão** — the default option. This type of crosswind diffusion reduces undershoots and overshoots to a minimum but can in rare cases give equation systems that are difficult to fully converge.
 - **Codina**. This option is less diffusive compared to the Do Carmo and Galeão option but can result in more undershoots and overshoots. It is also less effective for anisotropic meshes. The Codina option activates a text field for the **Lower gradient limit** g_{lim} . It defaults to $0.1 [\text{mol}/\text{m}^3] / \text{tds.helem}$, where tds.helem is the local element size.
- For both consistent stabilization methods, select an **Equation residual**. **Approximate residual** is the default and means that derivatives of the diffusion tensor components are neglected. This setting is usually accurate enough and is computationally faster. If required, select **Full residual** instead.

INCONSISTENT STABILIZATION


To display this section, click the **Show** button () and select **Stabilization**. By default, the **Isotropic diffusion** check box is not selected, because this type of stabilization adds artificial diffusion and affects the accuracy of the original problem. However, this option can be used to get a good initial guess for under resolved problems.

ADVANCED SETTINGS

To display this section, click the **Show** button () and select **Advanced Physics Options**. Normally these settings do not need to be changed. Select a **Convective term** — **Nonconservative form** (the default) or **Conservative form**. The conservative formulation

should be used for compressible flow. See [Convective Term Formulation](#) for more information.

DISCRETIZATION

To display all settings available in this section, click the **Show** button () and select **Advanced Physics Options**.

The **Compute boundary fluxes** check box is activated by default so that COMSOL Multiphysics computes predefined accurate boundary flux variables. When this option is checked, the solver computes variables storing accurate boundary fluxes from each boundary into the adjacent domain.

If the check box is cleared, the COMSOL Multiphysics software instead computes the flux variables from the dependent variables using extrapolation, which is less accurate in postprocessing results but does not create extra dependent variables on the boundaries for the fluxes.

The flux variables affected in the interface are:

- ndflux_c (where c is the dependent variable for the concentration). This is the normal diffusive flux and corresponds to the boundary flux when diffusion is the only contribution to the flux term.
- ntflux_c (where c is the dependent variable for the concentration). This is the normal total flux and corresponds to the boundary flux plus additional transport terms, for example, the convective flux when you use the nonconservative form.

Also the **Apply smoothing to boundary fluxes** check box is available if the previous check box is checked. The smoothing can provide a more well-behaved flux value close to singularities.



For details about the boundary fluxes settings, see [Computing Accurate Fluxes](#) in the *COMSOL Multiphysics Reference Manual*.

The **Value type when using splitting of complex variables** setting should in most pure mass transfer problems be set to **Real**, which is the default. It makes sure that the dependent variable does not get affected by small imaginary contributions, which can occur, for example, when combining a Time Dependent or Stationary study with a frequency-domain study. For more information, see [Splitting Complex-Valued Variables](#) in the *COMSOL Multiphysics Reference Manual*.



DEPENDENT VARIABLES

The dependent variable name is the **Concentration** c by default. The names must be unique with respect to all other dependent variables in the component.



Add or remove species variables in the model and also change the names of the dependent variables that represent the species concentrations.

Enter the **Number of species**. Use the **Add concentration** () and **Remove concentration** () buttons as needed.

FURTHER READING

	<ul style="list-style-type: none">• Theory for the Transport of Diluted Species Interface• Numerical Stabilization in the <i>COMSOL Multiphysics Reference Manual</i>.• In the <i>COMSOL Multiphysics Reference Manual</i>, see Table 2-4 for links to common sections and Table 2-5 for common feature nodes. You can also search for information: press F1 to open the Help window or Ctrl+F1 to open the Documentation window.
	<ul style="list-style-type: none">• <i>Effective Diffusivity in Porous Materials</i>: Application Library path COMSOL_Multiphysics/Diffusion/effective_diffusivity• <i>Micromixer</i>: Application Library path COMSOL_Multiphysics/Fluid_Dynamics/micromixer

The Transport of Diluted Species in Porous Media Interface

This interface (), found under the **Chemical Species Transport** branch (), is used to calculate the species concentration and transport in free and porous media. The interface is the same as the *Transport of Diluted Species* interface but it uses other defaults: The [Mass Transport in Porous Media](#) property is selected, and a [Porous Medium](#) node is added by default. The interface includes reaction rate expressions and solute sources for modeling of species transport and reaction in porous media.

This interface is dedicated to modeling transport in porous media, including immobile and mobile phases, where the chemical species may be subjected to diffusion, convection, migration, dispersion, adsorption, and volatilization in porous media. It

supports cases where either the solid phase substrate is exclusively immobile, or when a gas-filling medium is also assumed to be immobile.

It applies to one or more diluted species or solutes that move primarily within a fluid that fills (saturated) or partially fills (unsaturated) the voids in a solid porous medium. The pore space not filled with fluid contains an immobile gas phase. Models including a combination of porous media types can be studied.

The main feature nodes are the **Porous Medium** and **Unsaturated Porous Medium** nodes, which add the equations for the species concentrations and provide an interface for defining the properties of the porous media, as well as additional properties governing adsorption, volatilization, dispersion and diffusion, migration, and the velocity field to model convection.



The physics interface can be used for stationary and time-dependent analysis.

When this physics interface is added, these default nodes are also 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, reaction rate expressions, and species sources. You can also right-click **Transport of Diluted Species in Porous Media** to select physics features from the context menu.

SETTINGS

The rest of the settings are the same as [The Transport of Diluted Species Interface](#).

FURTHER READING

	<ul style="list-style-type: none">• Mass Balance Equation for Transport of Diluted Species in Porous Media• Domain, Boundary, and Pair Nodes for the Transport of Diluted Species Interface
	<ul style="list-style-type: none">• <i>Variably Saturated Flow and Transport — Sorbing Solute:</i> Application Library path Subsurface_Flow_Module/Solute_Transport/sorbing_solute Web link: https://www.comsol.com/model/variably-saturated-flow-and-transport-sorbing-solute-490

Domain, Boundary, and Pair Nodes for the Transport of Diluted Species Interface

The [Transport of Diluted Species Interface](#) has the following domain, boundary, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking to access the context menu (all users).



- To add a node, go to the **Physics** toolbar, no matter what operating system you are using.
 - Contextual subnodes (attributes) are available by clicking the parent node in the Model Builder, and then selecting the subnode from the **Attributes** menu.
-

- Adsorption
- Concentration
- Continuity on Interior Boundaries
- Dispersion
- Electrode Surface Coupling
- Equilibrium Reaction
- Fast Irreversible Surface Reaction
- Fluid
- Flux
- Flux Discontinuity
- Fracture
- Gas
- Inflow
- Initial Values
- Line Mass Source
- Liquid
- Mass-Based Concentrations
- No Flux
- Open Boundary
- Out-of-Plane Flux
- Outflow
- Partition Condition
- Periodic Condition
- Point Mass Source
- Porous Matrix
- Porous Medium
- Porous Electrode Coupling
- Reaction Coefficients
- Reactions
- Species Source
- Surface Reactions
- Surface Equilibrium Reaction
- Symmetry
- Thin Diffusion Barrier
- Thin Impermeable Barrier
- Transport Properties
- Unsaturated Porous Medium
- Volatilization

Some features require certain add-on modules. For details see <https://www.comsol.com/products/specifications/>



For axisymmetric components, COMSOL Multiphysics takes the axial symmetry boundaries (at $r = 0$) into account and automatically adds an **Axial Symmetry** node that is valid on boundaries representing the symmetry axis.



In the *COMSOL Multiphysics Reference Manual*, see [Table 2-4](#) for links to common sections and [Table 2-5](#) for common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Transport Properties

The settings in this node are dependent on the check boxes selected under [Transport Mechanisms](#) on the Settings window for the Transport of Diluted Species interface. It includes only the sections required by the activated transport mechanisms. It has all the equations defining transport of diluted species as well as inputs for the material properties.

MODEL INPUTS

The temperature model input is always available. Select the source of the **Temperature**. For **User defined**, enter a value or expression for the temperature in the input field. This input option is always available.

You can also select the temperature solved for by a Heat Transfer interface added to the model component. These physics interfaces are available for selection in the **Temperature** list.

CONVECTION

If transport by convection is active, the velocity field of the solvent needs to be specified. Select the source of the **Velocity field**. For **User defined**, enter values or expressions for the velocity components in the input fields. This input option is always available.

You can also select the velocity field solved for by a Fluid Flow interface added to the model component. These physics interfaces are available for selection in the **Velocity field** list.

DIFFUSION

Use this section to specify diffusion coefficients describing the diffusion of each species in the solvent fluid.

Use the **Source** list to select to pick up diffusion coefficients defined in a material or a Chemistry interface.

Select **Material**, and a **Fluid material** to use a diffusion coefficient in a material available in the model. User-defined property groups including a diffusion coefficient output property are available for selection in the **Fluid diffusion coefficient** list.

Select **Chemistry** to use a diffusion coefficient from a Chemistry interface added to model component. Diffusion coefficients are automatically defined when **Calculate transport properties** is selected on the interface level of a Chemistry interface. All defined diffusion coefficients are available for selection in the **Fluid diffusion coefficient** list.

For **User defined**, enter a value or expression for the **Fluid diffusion coefficient** D_c of each species in the corresponding input field. This can be a scalar value for isotropic diffusion or a tensor describing anisotropic diffusion. Select the appropriate tensor type — **Isotropic**, **Diagonal**, **Symmetric**, or **Full** that describes the diffusion transport, and then enter the values in the corresponding element (one value for each species).

Note that multiple species, as well as Migration in Electric fields (described below) is only available for certain COMSOL Multiphysics add-on products. See details: <https://www.comsol.com/products/specifications/>.

MIGRATION IN ELECTRIC FIELD

This section is available when the **Migration in electric field** check box is selected. From the **Electric potential** list, select the source of the electric field.

- Enter a value or expression for the **Electric potential** V , which is **User defined**; this input option is always available.
- Select the electric potential solved by an AC/DC-based interface that has also been added to the model.
- Select the electric potential defined or solved by Electrochemistry interface that has been added to the component.

By default the **Mobility** is set to be calculated based on the species diffusivity and the temperature using the **Nernst-Einstein relation**. For **User defined**, and under **Mobility**, select the appropriate scalar or tensor type — **Isotropic**, **Diagonal**, **Symmetric**, or **Full** — and type in the value of expression of the mobility $u_{m,c}$.

Enter the **Charge number** z_c (dimensionless, but requires a plus or minus sign) for each species.

The temperature (if you are using mobilities based on the Nernst–Einstein relation) is taken from **Model Inputs** section.

Note that the migration in electric fields feature is only available in some COMSOL products. See details: <https://www.comsol.com/products/specifications/>.

EXAMPLE MODELS

	<ul style="list-style-type: none"> • <i>Separation Through Dialysis</i>: Application Library path Chemical_Reaction_Engineering_Module/Mixing_and_Separation/dialysis Web link: https://www.comsol.com/model/separation-through-dialysis-258
	<ul style="list-style-type: none"> • <i>Transport in an Electrokinetic Valve</i>: Application Library path Microfluidics_Module/Fluid_Flow/electrokinetic_valve Web link: https://www.comsol.com/model/electrokinetic-valve-603

Initial Values

The **Initial Values** node specifies the initial values for the concentration of each species. These serve as an initial guess for a stationary solver or as initial conditions for a transient simulation.

DOMAIN SELECTION

If there are several types of domains with different initial values defined, it might be necessary to remove some domains from the selection. These are then defined in an additional **Initial Values** node.

INITIAL VALUES

Enter a value or expression for the initial value of the **Concentration** or concentrations, c_i . This also serves as a starting guess for stationary problems.

Mass-Based Concentrations

Use the **Mass-Based Concentrations** node to add postprocessing variables for mass-based concentrations (SI unit: kg/m^3) and mass fractions (dimensionless) for all species.

MIXTURE PROPERTIES

The default **Solvent density** ρ_{solvent} is taken **From material**. For **User defined**, enter a value or expression manually. Define the **Molar mass** of each species, which is needed to calculate the mass-based concentration.

Reactions

Use the **Reactions** node to account for the consumption or production of species through chemical reactions. Define the rate expressions as required.

DOMAIN SELECTION

From the **Selection** list, choose the domains on which to define rate expression or expressions that govern the source term in the transport equations.

Several reaction nodes can be used to account for different reactions in different parts for the modeling geometry.

REACTION RATES

Add a rate expression R_i (SI unit: $\text{mol}/(\text{m}^3 \cdot \text{s})$) for species i . Enter a value or expression in the field. Note that if you have the *Chemistry* interface available, provided with the *Chemical Reaction Engineering Module*, the reaction rate expressions can be automatically generated and picked up using the drop-down menu. For an example, see the application *Fine Chemical Production in a Plate Reactor* as linked below.

REACTING VOLUME

This section is only available when the **Mass Transport in Porous Media** property is available and selected. See <https://www.comsol.com/products/specifications/> for more details on availability.

When specifying reaction rates for a species in porous media, the specified reaction rate may have the basis of the total volume, the pore volume, or the volume of a particular phase.

- For **Total volume**, the reaction expressions in $\text{mol}/(\text{m}^3 \cdot \text{s})$ are specified per unit volume of the model domain (multiplied by unity).
- For **Pore volume**, the reaction expressions in $\text{mol}/(\text{m}^3 \cdot \text{s})$ are specified per unit volume of total pore space. The reaction expressions will be multiplied by the domain porosity, ϵ_p . (ϵ_p equals unity for nonporous domains.)

- For **Liquid phase**, the reaction expressions in $\text{mol}/(\text{m}^3 \cdot \text{s})$ are specified per unit volume of liquid in the pore space. The expressions will be multiplied by the liquid volume fraction θ . (θ equals ϵ_p for Saturated Porous Media domains).
- For **Gas phase**, the expressions are multiplied by the gas volume fraction $\alpha_v = \epsilon_p - \theta$. α_v equals 0 for Saturated Porous Media domains.

FURTHER READING

See the theory chapter on chemical species transport, starting with the section *Mass Balance Equation*.



- *Fine Chemical Production in a Plate Reactor*: Application Library path
Chemical_Reaction_Engineering_Module/Reactors_with_Mass_and_Heat_Transfer/plate_reactor
Web link:
<https://www.comsol.com/model/fine-chemical-production-in-a-plate-reactor-8589>

No Flux

This node is the default boundary condition on exterior boundaries. It should be used on boundaries across which there is no mass flux, typically solid walls where no surface reactions occur. The condition applied for each species corresponds to

$$-\mathbf{n} \cdot (-D \nabla c) = 0$$

where \mathbf{n} denotes the outward pointing normal of the boundary. When the mass transport includes migration of ionic species, the no flux condition is:

$$-\mathbf{n} \cdot (-D \nabla c - z u_m F c \nabla \phi) = 0$$

CONVECTION

By default, the feature prescribes a vanishing flux due to diffusion and migration in an electric field. This is the appropriate no flux condition when the relative convective velocity at the boundary is zero. When the fluid velocity at the boundary is not equal to that of the boundary, it is often convenient to prescribe the total flux including the convection. To do this select **Include** in the **Convection** section.

When including the convection, the no flux condition prescribed is

$$-\mathbf{n} \cdot (-D\nabla c + \mathbf{u}c) = 0$$

or

$$-\mathbf{n} \cdot (-D\nabla c - zu_m Fc\nabla\phi + \mathbf{u}c) = 0$$

when migration of ionic species is included.

Inflow

Use this node to specify all species concentrations at an inlet boundary.

If you want to specify the concentration of a subset of the partaking species, this can be done by using the [Concentration](#) node instead.

For the **Electroanalysis** interface, this node is available when you select the **Convection** check box on the physics interface **Settings** window.

CONCENTRATION


For the concentration of each species $c_{0,c}$ (SI unit: mol/m³), enter a value or expression.

BOUNDARY CONDITION TYPE

The option **Concentration constraint** constrains the concentration values on the boundary by the use of pointwise constraints.

The other option, **Flux (Danckwerts)** can be used when the concentration at the boundary is not known, or when it varies in a non-trivial manner. This may, for example, be useful when reactions with high reaction rates occur in the vicinity of the inlet. In this case the concentration far upstream of the boundary is instead prescribed. The **Flux (Danckwerts)** condition prescribes the total flux defined by the upstream concentration and the fluid velocity at the boundary.

CONSTRAINT SETTINGS

To display this section, click the **Show** button () and select **Advanced Physics Options**. You can find details about the different constraint settings in the section [Constraint Reaction Terms](#) in the *COMSOL Multiphysics Reference Manual*.

FURTHER READING

See the theory chapter in the section [Danckwerts Inflow Boundary Condition](#).

Outflow

Apply this condition at outlets boundaries where species should be transported out of the model domain by fluid motion or by an electric field (in the case of ions). It is assumed that convection and migration (in an electric field) are the dominating transport mechanisms across the boundary, and therefore that the diffusive transport can be ignored, that is:

$$\mathbf{n} \cdot (-D\nabla c) = 0$$

Note that the **Convection** or the **Migration in electric field** transport mechanisms needs to be included for this node to be available.


Concentration

This condition node adds a boundary condition for the species concentration. For example, a $c = c_0$ condition specifies the concentration of species c .

CONCENTRATION

Individually specify the concentration for each species. Select the check box for the **Species** to specify the concentration, and then enter a value or expression in the corresponding field. To use another boundary condition for a specific species, click to clear the check box for the concentration of that species.

CONSTRAINT SETTINGS

To display this section, click the **Show** button () and select **Advanced Physics Options**. You can find details about the different constraint settings in the section [Constraint Reaction Terms](#) in the *COMSOL Multiphysics Reference Manual*.

Flux

This node can be used to specify the species molar flux across a boundary. The flux can for example occur due to chemical reactions or a phase change at the boundary. The flux can also represent the transport to or from a surrounding environment currently not included model.

The prescribed flux of a species c is by default defined as

$$-\mathbf{n} \cdot (-D\nabla c) = J_0$$

where \mathbf{n} denotes the outward pointing normal of the boundary. When the mass transport includes migration of ionic species, the flux is defined as:

$$-\mathbf{n} \cdot (-D\nabla c - zu_m Fc \nabla \phi) = J_0$$

The flux prescribed, J_0 , can include any arbitrary user-specified expression. It can be constant or a function of a dependent variable or independent variable. Common examples are a flux dependent of the concentration, temperature, pressure or the electric potential ϕ .

CONVECTION

By default, the flux due to diffusion and migration in an electric field is prescribed. This is the appropriate flux condition when the relative velocity at the boundary is zero. When the fluid velocity is not equal to that of the boundary, it is often convenient to prescribe the total flux, including the convection. To do this select **Include** in the **Convection** section.

When including the convection, the prescribed flux is defined as:

$$-\mathbf{n} \cdot (-D\nabla c + \mathbf{u}c) = J_0$$

or

$$-\mathbf{n} \cdot (-D\nabla c - zu_m Fc \nabla \phi + \mathbf{u}c) = J_0$$

when migration of ionic species is included.

INWARD FLUX

Select the Species check box for the species for which to specify the flux, and enter a value or expression for the inward flux in the corresponding field. Use a minus sign when specifying a flux directed out of the system. To use another boundary condition for a specific species, click to clear the check box for that species.

External convection

Set **Flux type** to **External convection** to prescribe a flux to or from an exterior domain (not modeled) assumed to include convection. The exterior can for example include a forced convection to control the temperature or to increase the mass transport. In this case the prescribed mass flux corresponds to

$$J_0 = k_c(c_b - c)$$

where k_c is a mass transfer coefficient and c_b is the bulk concentration, the typical concentration far into the surrounding exterior domain.

Symmetry

The **Symmetry** node can be used to represent boundaries where the species concentration is symmetric, that is, where there is no mass flux across the boundary.

This boundary condition is identical to that of the [No Flux](#) node.

Flux Discontinuity

This node represents a discontinuity in the mass flux across an interior boundary:

$$-\mathbf{n} \cdot [(\mathbf{J} + \mathbf{u}c)_u - (\mathbf{J} + \mathbf{u}c)_d] = N_0 \quad \mathbf{J} = -D\nabla c$$

where the value N_0 (SI unit: $\text{mol}/(\text{m}^2 \cdot \text{s})$) specifies the jump in total flux at the boundary. This can be used to model a boundary source, for example a surface reaction, adsorption or desorption.

FLUX DISCONTINUITY

In this section the jump in species flux (or surface source) is specified.

Select the Species check box for the species to specify and enter a value or expression for the material flux jump in the corresponding field. To use a different boundary condition for a specific species, click to clear the check box for the flux discontinuity of that species.

Partition Condition

The **Partition Condition** node can be used to prescribe the relation between the concentration of a solute species in two adjoining immiscible phases. It can for example be used on interior boundaries separating two liquid phases, a gas-liquid interface, or on a boundary separating a liquid phase and a solid or porous media.

Select **Partition coefficient** from the **Relation** list to prescribe a linear relationship between the concentration on either side of a boundary. For a species concentration c_i , the ratio between the concentration on the upside and on the downside of the boundary ($c_{i,u}$ and $c_{i,d}$ respectively) is then defined in terms of a partition coefficient K_i in the manner of

$$K_i = \frac{c_{i,u}}{c_{i,d}}$$

Use the associated input field to prescribe the partition coefficient for each species.

Select **User defined** from the **Relation** list to set up a nonlinear relation between the up and downside concentrations. For such relations $f(c_i)_u = f(c_i)_d$, where $f(c_i)_u$ is the expression enforced on the upside. Use the associated input fields to enter the upside and downside expressions for each species.

The up and downside of the selected boundary is indicated with a red arrow in the Graphics window. The arrow points from the downside into the upside. Select the **Reverse direction** check box to reverse the direction of the arrow on the selected boundary, and update the definition of the up and downside concentrations accordingly.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

FURTHER READING

For an example of using a partition condition, see this application example:



Separation Through Dialysis: Application Library path
Chemical_Reaction_Engineering_Module/Mixing_and_Separation/dialysis

Periodic Condition

The **Periodic Condition** node can be used to define periodicity for the mass transport between two sets of boundaries. The node prescribes continuity in the concentration and the mass flux between the “source” and the “destination” side respectively. Note that these names are arbitrary and does not influence the direction in which mass is transported. It is dictated by mass transfer equations in the adjacent domains.

The node can be activated on more than two boundaries, in which case the feature tries to identify two separate surfaces that each consist of one or several connected boundaries.

For more complex geometries, it might be necessary to add the **Destination Selection** subnode, which is available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu. With this subnode, the boundaries that constitute the source and destination surfaces can be manually specified.

FURTHER READING

For an example of using a periodic condition, see this application example:



The KdV Equation and Solitons: Application Library path
COMSOL_Multiphysics/Equation_Based/kdv_equation

Line Mass Source

The **Line Mass Source** feature models mass flow originating from a tube or line region with an infinitely small radius.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

SELECTION

The **Line Mass Source** feature is available for all dimensions, but the applicable selection differs between the dimensions.

MODEL DIMENSION	APPLICABLE GEOMETRICAL ENTITY
2D	Points
2D Axisymmetry	Points not on the symmetry axis and the symmetry axis
3D	Edges

SPECIES SOURCE

Enter the source strength, $\dot{q}_{l,c}$, for each species (SI unit: mol/(m·s)). A positive value results in species injection from the line into the computational domain, and a negative value means that the species is removed from the computational domain.

Line sources located on a boundary affect the adjacent computational domains. This effect makes the physical strength of a line source located in a symmetry plane twice the given strength.

FURTHER READING

See the section [Mass Sources for Species Transport](#).

Point Mass Source

The **Point Mass Source** feature models mass flow originating from an infinitely small domain around a point.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

SPECIES SOURCE

Enter the source strength, $\dot{q}_{p,c}$, for each species (SI unit: mol/s). A positive value results in species injection from the point into the computational domain, and a negative value means that the species is removed from the computational domain.

Point sources located on a boundary or on an edge affect the adjacent computational domains. This has the effect, for example, that the physical strength of a point source located in a symmetry plane is twice the given strength.

FURTHER READING

See the section [Mass Sources for Species Transport](#).

Open Boundary

Use this node to set up mass transport across boundaries where both convective inflow and outflow can occur. On the parts of the boundary where fluid flows into the domain, an exterior species concentration is prescribed. On the remaining parts, where fluid flows out of the domain, a condition equivalent to the **Outflow** node is instead prescribed.

The direction of the flow across the boundary is typically calculated by a fluid flow interface and is provided as a model input to the Transport of Diluted Species interface.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

EXTERIOR CONCENTRATION

Enter a value or expression for the **Exterior concentration**.

Thin Diffusion Barrier

Use this boundary condition to model a thin layer through which mass is transported by diffusion only. The node is applicable on interior boundaries and can be used to avoid meshing thin structures.

THIN DIFFUSION BARRIER

Specify the **Layer thickness**, d_s , and input a **Diffusion coefficient**, $D_{s,c}$, for each of the species included.

Thin Impermeable Barrier

This feature models a thin mass transfer barrier. It is available on interior boundaries and introduces a discontinuity in the concentration across the boundary. On each side, a no-flux condition is prescribed for the mass transport implying that it acts as a barrier. The feature can be used to avoid meshing thin structures.

Solving a model involving coupled fluid flow and mass transfer, the Thin Impermeable Barrier feature can be combined with an Interior Wall feature in order to model a thin solid wall.

Equilibrium Reaction

Use this node to model a reaction where the kinetics is so fast that the equilibrium condition is fulfilled at all times. The node solves for an additional degree of freedom (the reaction rate R_{eq}) to fulfill the equilibrium condition at all times in all space coordinates.

If the **Apply equilibrium condition on inflow boundaries** check box is selected, the specified inflow concentration values in all active **Inflow** boundary nodes for the physics interface are modified to comply with the equilibrium condition.



- A necessary requirement for this feature to be available is that two or more species are solved for by the interface.
- This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

EQUILIBRIUM CONDITION

The list defaults to **Equilibrium constant** or select **User defined**. For either option, the **Apply equilibrium condition on inflow boundaries** check box is selected by default.

For **Equilibrium constant**, enter an **Equilibrium constant** K_{eq} (dimensionless). Also enter a value or expression for the **Unit activity concentration** C_{a0} (SI unit: mol/m³).

Selecting **Equilibrium constant** defines an equilibrium condition based on the stoichiometric coefficients, the species activities, and the law of mass action.

For **User defined**, enter an **Equilibrium expression** E_{eq} (dimensionless).

STOICHIOMETRIC COEFFICIENTS

Enter a value for the stoichiometric coefficient ν_c (dimensionless). The default is 0. Use negative values for reactants and positive values for products in the modeled reaction.

Species with a stoichiometric coefficient value of 0 are not affected by the **Equilibrium Reaction** node.

Surface Reactions

The **Surface Reactions** node can be used to account for the species boundary flux due to chemical reactions occurring on a surface (heterogeneous reactions). For a domain species participating in a surface reaction, the boundary flux corresponds to the reaction rate at the surface.

SURFACE REACTION RATE

Specify the surface reaction rate J_0 of each species resulting from the reactions. Note that if you have the Chemistry interface available, provided with the Chemical

Reaction Engineering Module, the reaction rate expressions can be automatically generated and picked up using the drop-down menu.

FURTHER READING

For an example of using the Surface Reactions node, see this application example:



Chemical Vapor Deposition of GaAs: Application Library path
Chemical_Reaction_Engineering_Module/Reactors_with_Mass_and_Heat_Transfer/gaas_cvd

Surface Equilibrium Reaction

Use this node to model an equilibrium reaction on a boundary (surface). The settings for this node are similar to [Equilibrium Reaction](#). Note that a necessary requirement for this feature to be available is that two or more species are solved for by the interface.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

Fast Irreversible Surface Reaction

This boundary node defines an irreversible reaction where the kinetics is so fast that the only factor limiting the reaction rate is the transport of a species to the reacting surface.

The node will set the **Rate limiting species concentration to zero** at the boundary, and balance the fluxes of the species participating in the reaction and the current densities according to the Stoichiometric Coefficients settings.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

Porous Electrode Coupling

Use this node to add a molar source in a domain that is coupled to one or multiple [Porous Electrode Reaction](#) nodes of an Electrochemistry Interface.

The molar source is calculated from the number of electrons, stoichiometric coefficients, and volumetric current densities of the coupled porous electrode reactions specified in the **Reaction Coefficients** subnodes.

In the Transport of Concentrated Species interface, the molar sources (or sinks) are multiplied by the species molar masses to obtain the corresponding mass sources.

Additional [Reaction Coefficients](#) subnodes are available from the context menu (right-click the parent node) as well as from the **Physics** toolbar, **Attributes** menu.

Note that if you are also modeling the momentum transport and expect a non-negligible total mass source or sink, which is often the case in gas diffusion electrodes, you need to also add a corresponding Porous Electrode Coupling node in the Fluid Flow interface.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

Reaction Coefficients

Add this node to the [Electrode Surface Coupling](#) and [Porous Electrode Coupling](#) features to define molar fluxes and sources based on electrode current densities in an **Electrochemistry** interface.

The molar flux or source is proportional to the stoichiometric coefficients and the current density according to Faraday's law.

Current densities from **Electrode Reaction** (i_{loc} , SI unit: A/m²) or **Porous Electrode Reaction** nodes (i_v , SI unit: A/m³) of any **Electrochemistry** interface in the model are available for selection as the **Coupled reaction**, and user-defined expressions are also supported.

Enter the **Number of participating electrons** n_m (dimensionless) and the **Stoichiometric coefficient** ν_c (dimensionless) as explained in the theory section linked below.

Use multiple subnodes to couple to multiple reactions.

Electrode Surface Coupling

Use this node to define a flux boundary condition based on current densities of one or multiple [Electrode Reaction](#) nodes in an Electrochemistry interface.

The flux is proportional to the current densities and the stoichiometric coefficients according to Faraday's law as defined by summation over the [Reaction Coefficients](#) subnodes.

Note that if you are also modeling the momentum transport and expect a nonnegligible total mass flux over the boundary, which is often the case for gas diffusion electrodes, you need to also add a corresponding Electrode Surface Coupling node in the Fluid Flow interface.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

Porous Medium

Use this node to model the concentration of diluted species transported by a solvent (mobile fluid phase) through interstices in a solid porous medium. In addition to transport due to convection and diffusion, the node contains functionality to include species evolution through adsorption and dispersion.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

Species transport through a porous medium is affected both by properties of the fluid phase, and as well as properties of the solid matrix. These properties are defined in the [Fluid](#) and [Porous Matrix](#) subnodes respectively. The Porous Medium node supports material properties using a [Porous Material](#) node (described in the *COMSOL Multiphysics Reference Manual*), where properties are defined per phase in a similar manner.

Add an [Adsorption](#) or a [Dispersion](#) subfeature to the **Porous Medium** to account for the corresponding transport mechanism.

Fluid

Use this node to specify the mass transfer in the mobile fluid solvent filling the pores of a porous medium. It is used as a subnode to [Porous Medium](#).

MODEL INPUTS

The temperature model input is always available. By default, the **Temperature** model input is set to **Common model input**, and the temperature is controlled from [Default Model Inputs](#) under **Global Definitions** or by a locally defined [Model Input](#). If a Heat Transfer interface is included in the component, it controls the temperature **Common model input**. Alternatively, the temperature field can be selected from another physics interface. All physics interfaces have their own tags (**Name**). For example, if a Heat Transfer in Fluids interface is included in the component, the **Temperature (ht)** option is available for T .

You can also select **User defined** from the **Temperature** model input in order to manually prescribe T .

CONVECTION

If transport by convection is active, the velocity field of the solvent needs to be specified. For **User defined**, enter values or expressions for the velocity components in the input fields. This input option is always available. You can also select the velocity field solved for by a Fluid Flow interface added to the model component. These physics interfaces are available for selection in the **Velocity field** list.

DIFFUSION

Use this section to specify diffusion coefficients describing the diffusion of each species in the solvent fluid.

Use the **Source** list to select to pick up diffusion coefficients defined in a material or a Chemistry interface.

Select **Material**, and a **Fluid material** to use a diffusion coefficient in a material available in the model. User-defined property groups including a diffusion coefficient output property are available for selection in the **Fluid diffusion coefficient** list.

Select **Chemistry** to use a diffusion coefficient from a Chemistry interface added to model component. Diffusion coefficients are automatically defined when **Calculate transport properties** is selected on the interface level of a Chemistry interface. All defined diffusion coefficients are available for selection in the **Fluid diffusion coefficient** list.

For **User defined**, enter a value or expression for the **Fluid diffusion coefficient** $D_{F,i}$ of each species in the corresponding input field.

In a porous medium the diffusivity is reduced due to the fact that the solid grains impede Brownian motion. Select an **Effective diffusivity model** to account for the reduced diffusivity. The available models are **Millington and Quirk model** (the default), **Bruggeman model**, **Tortuosity model**, or **No correction**. For **Tortuosity model**, enter a value for the tortuosity $\tau_{F,i}$ (dimensionless).

Note that multiple species, as well as Migration in Electric fields (described below) is only available for certain COMSOL Multiphysics add-on products. See details: <https://www.comsol.com/products/specifications/>.

MIGRATION IN ELECTRIC FIELD

This section is available when the **Migration in electric field** check box is selected. From the **Electric potential** list, select the source of the electric field.

- For **User defined**, enter a value or expression for the **Electric potential** V . This input option is always available.
- Select the electric potential solved by an AC/DC-based interface that has added to the component.
- Select the electric potential defined or solved for by an Electrochemistry interface added to the component.

By default the **Mobility** is set to be calculated based on the species effective diffusivity and the temperature using the **Nernst-Einstein relation**. For **User defined**, select the appropriate scalar or tensor type — **Isotropic**, **Diagonal**, **Symmetric**, or **Full** — and type in the value or expression of the effective mobility $u_{me,i}$.

Enter the **Charge number** z_c for each species.

Porous Matrix

This node sets the porosity when modeling transport of diluted species in a **Porous Medium** or a **Unsaturated Porous Medium**.

The default **Porosity** ϵ_p of the solid matrix is taken **From material**. The Porous Matrix node supports the use of a **Porous Material** node, where the porosity is defined in the manner of

$$\varepsilon_p = 1 - \sum_i \theta_{s,i} - \sum_i \theta_{\text{imf},i}$$

where $\theta_{s,i}$ and $\theta_{\text{imf},i}$ are the porosities of the **Solid** and **Immobile Fluids** subnodes under the **Porous Material** node.

Select **From pellet bed densities** to compute the porosity using the (dry bulk) **Bed density** ρ_b and the (single phase) **Pellet density** ρ_{pe} . The porosity is then defined from

$$\varepsilon_p = 1 - \frac{\rho_b}{\rho_{pe}}$$

For **User defined**, enter a value or expression for the porosity.

Dispersion

Local variations in the velocity as the fluid flows around solid particles lead to mechanical mixing, referred to as dispersion. Use this feature to account for dispersion in a [Porous Medium](#) or a [Unsaturated Porous Medium](#)

This subfeature is available when both the **Mass transfer in porous media mass transfer** check box and the **Convection** check box are selected on the **Settings** window for the physics interface.

Select the **Specify dispersion for each species individually** check box to specify the dispersion tensor D_D (SI unit: m^2/s) for each species separately. When not selected the same dispersion tensor D_D is used for all species.

Select an option from the **Dispersion tensor** list — **Dispersivity** or **User defined**.

Select **Dispersivity** to specify the dispersion in terms of dispersivities (SI unit: m). Select an option from the **Dispersivity model** list: **Isotropic** (the default) or **Transverse isotropic** based on the properties of the porous media. For isotropic porous media, specify the longitudinal and transverse dispersivities. For transverse isotropic porous media, specify the longitudinal, horizontal transverse, and vertical transverse dispersivities.

For **User defined**, specify the dispersion components in terms of constants or expressions. Select **Isotropic**, **Diagonal**, **Symmetric**, or **Full** to enable the appropriate tensor components.

Unsaturated Porous Medium

Use this node to model the concentration of diluted species transported by a solvent (mobile fluid phase) through interstices in a solid porous medium. In addition to transport due to convection and diffusion, the node contains functionality to include species evolution through adsorption and dispersion.

Use this node to model the concentration of diluted species transported by a liquid (mobile fluid phase) in a partially filled solid porous medium. The interstices of the porous medium contains the liquid carrier phase and gas pockets. Apart from convection and diffusion, the node contains functionality to include species evolution through adsorption, dispersion, and volatilization.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

The properties of each phase present are defined using the [Liquid](#), the [Gas](#), and the [Porous Matrix](#) subnodes respectively. The Unsaturated Porous Medium node supports material properties using a [Porous Material](#) node (described in the *COMSOL Multiphysics Reference Manual*), where properties are defined per phase in a similar manner.

Add an [Adsorption](#) or a [Dispersion](#) subfeature to the **Unsaturated Porous Medium** to account for the corresponding transport mechanism.

Liquid

Use this node to specify the mass transfer in the mobile liquid solvent present in the pores of the [Unsaturated Porous Medium](#).

MODEL INPUTS

The temperature model input is always available. By default, the **Temperature** model input is set to **Common model input**, and the temperature is controlled from [Default Model Inputs](#) under **Global Definitions** or by a locally defined [Model Input](#). If a Heat Transfer interface is included in the component, it controls the temperature **Common model input**. Alternatively, the temperature field can be selected from another physics interface. All physics interfaces have their own tags (**Name**). For example, if a Heat Transfer in Fluids interface is included in the component, the **Temperature (ht)** option is available for T .

You can also select **User defined** from the **Temperature** model input in order to manually prescribe T .

SATURATION

Select **Saturation** or **Liquid volume fraction** from the list.

For **Saturation**, enter a value for s (dimensionless) between 0 and 1. The liquid volume fraction is then computed from the saturation and porosity as $\theta_l = \varepsilon_p s$.

For **Liquid volume fraction**, enter a value for θ_l (dimensionless) between 0 and the value of the porosity.

Select a **Liquid fraction time change**: **Liquid fraction constant in time** (the default), **Time change in liquid fraction**, or **Time change in pressure head**.

- For **Time change in fluid fraction**, enter $d\theta/dt$ (SI unit: 1/s).
- For **Time change in pressure head**, enter dH_p/dt (SI unit: m/s) and a **Specific moisture capacity** C_m (SI unit: 1/m). If a Darcy's Law interface is included in the component, the time change in pressure head solved for can be selected.

CONVECTION

If transport by convection is active, the velocity field of the solvent needs to be specified. For **User defined**, enter values or expressions for the velocity components in the input fields. This input option is always available. You can also select the velocity field solved for by a Fluid Flow interface added to the model component. These physics interfaces are available for selection in the **Velocity field** list.

DIFFUSION

Use this section to specify diffusion coefficients describing the diffusion of each species in the solvent liquid.

Use the **Source** list to select to pick up diffusion coefficients defined in a material or a Chemistry interface.

Select **Material**, and a **Liquid material** to use a diffusion coefficient in a material available in the model. User-defined property groups including a diffusion coefficient output property are available for selection in the **Liquid diffusion coefficient** list.

Select **Chemistry** to use a diffusion coefficient from a Chemistry interface added to model component. Diffusion coefficients are automatically defined when **Calculate transport properties** is selected on the interface level of a Chemistry interface. All

defined diffusion coefficients are available for selection in the **Liquid diffusion coefficient** list.

For **User defined**, enter a value or expression for the **Liquid diffusion coefficient** $D_{L,i}$ of each species in the corresponding input field.

In a porous medium the diffusivity is reduced due to the fact that the solid grains impede Brownian motion. Select an **Effective diffusivity model, liquid** to account for the reduced diffusivity in the liquid. The available models are **Millington and Quirk model** (the default), **Bruggeman model**, **Tortuosity model**, or **No correction**. For **Tortuosity model**, enter a value for the tortuosity $\tau_{L,i}$ (dimensionless).

Note that multiple species, as well as Migration in Electric fields (described below) is only available for certain COMSOL Multiphysics add-on products. For details see: <https://www.comsol.com/products/specifications/>.

MIGRATION IN ELECTRIC FIELD

This section is available when the **Migration in electric field** check box is selected. From the **Electric potential** list, select the source of the electric field.

- For **User defined**, enter a value or expression for the **Electric potential** V . This input option is always available.
- Select the electric potential solved by an AC/DC-based interface that has added to the component.
- Select the electric potential defined or solved for by an Electrochemistry interface added to the component.

By default the **Mobility** is set to be calculated based on the species effective diffusivity and the temperature using the **Nernst-Einstein relation**. For **User defined**, select the appropriate scalar or tensor type — **Isotropic**, **Diagonal**, **Symmetric**, or **Full** — and type in the value or expression of the effective mobility $u_{me,i}$.

Enter the **Charge number** z_c for each species.

Gas

Use this node to specify the mass transfer in the gas phase present in the pores of the **Unsaturated Porous Medium**.

DIFFUSION

The settings for the diffusion of each species in the gas phase are identical to those in the [Liquid](#) subnode.

VOLATILIZATION

Enter a value for the volatilization isotherm $k_{G,c}$ (dimensionless) for each species.

Adsorption

Use this node to model adsorption of the (fluid phase) solute species onto the surface of the porous matrix. It is available as a subnode to the [Porous Medium](#) and the [Unsaturated Porous Medium](#) nodes.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

MATRIX PROPERTIES

The density of the porous media is needed when modeling adsorption to the surface of the porous matrix. Choose to input either the **Dry bulk density** ρ , or the **Solid phase density** ρ_s . The former is the density of the porous matrix including empty pores, while the latter corresponds to the density of the pure solid phase. The density can be defined from the domain material by selecting **From material**, or from a user defined expression. When a **Porous Material** is used on the selection, the density will be requested from a **Solid** subfeature. Several **Solid** features can be used to model a homogeneous mixture of several solid components.

ADSORPTION

Select an **Adsorption isotherm** — **Langmuir**, **Freundlich**, **Toth**, **BET**, or **User defined** to specify how to compute c_p , the amount of species sorbed to the solid phase (moles per unit dry weight of the solid):

- For **Langmuir**:

$$c_P = c_{P\max} \frac{K_L c}{1 + K_L c} \quad , \quad K_P = \frac{\partial c_P}{\partial c} = \frac{K_L c_{P\max}}{(1 + K_L c)^2}$$

Enter a **Langmuir constant** $k_{L,c}$ (SI unit: m^3/mol) and an **Adsorption maximum** $c_{p,\max,c}$ (SI unit: mol/kg).

- For **Freundlich**:

$$c_P = K_F \left(\frac{c}{c_{\text{ref}}} \right)^N \quad , \quad K_P = \frac{\partial c_P}{\partial c} = N \frac{c_P}{c}$$

Enter a **Freundlich constant** $k_{F,c}$ (SI unit: mol/kg), a **Freundlich exponent** $N_{F,c}$ (dimensionless), and a **Reference concentration** $c_{\text{ref},c}$ (SI unit: mol/m^3).

- For **Toth**:

$$c_P = c_{P\max} \frac{b_T c}{(1 + (b_T c)^{N_T})^{1/N_T}}$$

and

$$K_P = \frac{\partial c_P}{\partial c} = c_{P\max} b_T (1 + (b_T c)^{N_T})^{-\left(1 + \frac{1}{N_T}\right)}$$

Enter a **Toth constant** $b_{T,c}$ (SI unit: m^3/mol), a **Toth exponent** $N_{T,c}$ (dimensionless), and an **Adsorption maximum** $c_{p,\max,c}$ (SI unit: mol/kg).

- For **BET (Brunauer-Emmett-Teller)**:

$$c_P = \frac{K_B c_0 c}{(c_S - c) \left(1 + (K_B - 1) \frac{c}{c_S} \right)}$$

and

$$K_P = \frac{\partial c_P}{\partial c} = \frac{K_B c_0 c_S ((K_B - 1) c^2 + c_S^2)}{(c - c_S)^2 (c_S + (K_B - 1) c)^2}$$

Enter a **BET constant** $K_{B,c}$ (dimensionless), a **Monolayer adsorption capacity** $c_{0,c}$ (SI unit: mol/kg), and an **Saturation concentration** $c_{S,c}$ (SI unit: mol/m^3).

- For **User defined** enter an **Adsorption isotherm** $c_{P,c}$ (SI unit: mol/kg):

$$c_p = f(c)$$

For more information, see [Adsorption](#) in the theory section.

FURTHER READING

See the theory chapter in the section [Mass Balance Equation for Transport of Diluted Species in Porous Media](#).

Volatilization

This feature is available when the **Mass transfer in porous media** check box is selected on the **Settings** window for the physics interface.

Use this feature to model mass transfer at the boundary due to volatilization. The species dissolved in the liquid are assumed to be vaporized at the boundary, and transported into the surrounding bulk region due to convection and diffusion. The mass transfer at the boundary is defined as

$$-\mathbf{n} \cdot \mathbf{J}_c = -h_c(k_{G,c}c - c_{\text{Gatm},c})$$

where h_c is the mass transfer coefficient, $k_{G,c}$ the volatilization coefficient, and $c_{\text{Gatm},c}$ the concentration in the surrounding atmosphere.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

VOLATILIZATION

Enter a **Mass transfer coefficient** h_c defining the transfer into the surrounding media. This can be given by boundary layer theory. When assuming that no convective flow is present in the surrounding, the mass transfer coefficient can be defined from the gas diffusion coefficient D_{Gc} and the thickness of the diffusion layer d_s in the manner of

$$h_c = \frac{D_{Gc}}{d_s}$$

Also give the atmospheric concentration for each species, $c_{\text{Gatm},c}$. The **Volatilization** coefficient $k_{G,c}$ for each species are taken from the adjacent [Unsaturated Porous Medium](#) domain.

Species Source

In order to account for consumption or production of species in porous domains, the **Species Source** node adds source terms expressions S_i to the right-hand side of the species transport equations.

DOMAIN SELECTION


From the **Selection** list, choose the domains on which to define rate expression or expressions that govern the source term in the transport equations.

If there are several types of domains, with subsequent and different reactions occurring within them, it might be necessary to remove some domains from the selection. These are then defined in an additional **Species Source** node.

SPECIES SOURCE

Add a source term S_i (SI unit: $\text{mol}/(\text{m}^3 \cdot \text{s})$) for each of the species solved for. Enter a value or expression in the field of the corresponding species.

Hygroscopic Swelling

The **Hygroscopic Swelling** multiphysics coupling node () is used for moisture concentration coupling between the Solid Mechanics interface and either the Transport of Diluted Species or Transport of Diluted Species in Porous Media interfaces.

Hygroscopic swelling is an effect of internal strain caused by changes in moisture content. This volumetric strain can be written as

$$\epsilon_{\text{hs}} = \beta_{\text{h}} M_{\text{m}} (c_{\text{mo}} - c_{\text{mo,ref}})$$

where β_{h} is the coefficient of hygroscopic swelling, M_{m} is the molar mass, c_{mo} is the moisture concentration, and $c_{\text{mo,ref}}$ is the strain-free reference concentration.

This feature requires a license of either the MEMS Module or the Structural Mechanics Module. The multiphysics feature will appear automatically if both the Transport of Diluted Species and the Solid Mechanics interfaces are added to the same component. For the most current information about licensing, please see See <https://www.comsol.com/products/specifications/>.

FURTHER READING

More information about how to use hygroscopic swelling can be found in [Hygroscopic Swelling Coupling](#) section in the *Structural Mechanics Module User's Guide*.

More information about multiphysics coupling nodes can be found in the section [The Multiphysics Branch](#) in the *COMSOL Multiphysics Reference Manual*.

Out-of-Plane Flux

The out-of-plane flux node is used to prescribe a flux in an out-of-plane domain, and it can be used to reduce a model geometry to 2D, or even 1D, when the concentration variation is small in one or more directions. This could be the case for example when the object to model is thin or slender. [Figure 5-1](#) shows examples of possible situations in which this type of geometry reduction can be applied.

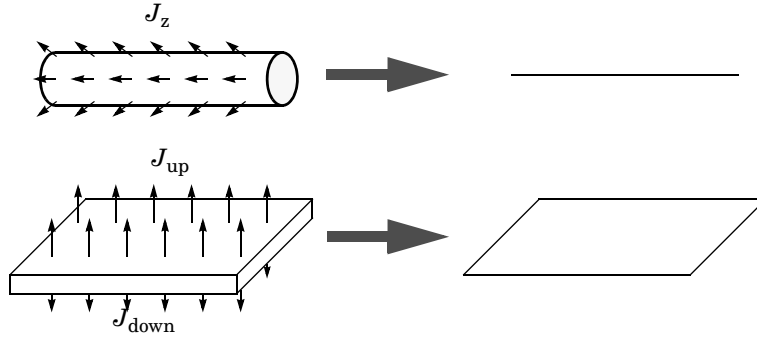


Figure 5-1: Geometry reduction from 3D to 1D (top) and from 3D to 2D (bottom).

For a 1D component this node adds a single out-of-plane molar flux $J_{0,z,i}$ for species i . For a 2D component two fluxes can be prescribed for each species; $J_{0,u,i}$ for the upside of the domain, and $J_{0,d,i}$ for the downside of the domain.

The fluxes are added as a source contribution to the right-hand side of the transport equation. For example, in 2D components with diffusion, convection and out-of-plane flux present, mass transfer of species i is defined by the mass conservation equation

$$\frac{\partial c_i}{\partial t} + \nabla \cdot \mathbf{J}_i + \mathbf{u} \cdot \nabla c_i = R_i + S_{\text{opf},i} \quad , \quad (5-1)$$

where $S_{\text{opf},i}$ is the out-of-plane source for species i

$$S_{\text{opf},i} = \frac{J_{0,i}}{d_z} = \frac{J_{0,\text{u},i} + J_{0,\text{d},i}}{d_z}.$$

For external convection on the upside and the downside of the domain, the out-of-plane flux is

$$J_{0,i} = k_{c,\text{u},i}(c_{b,\text{u},i} - c_i) + k_{c,\text{d},i}(c_{b,\text{d},i} - c_i)$$

where k_c is a mass transfer coefficient and c_b is the bulk concentration, the typical concentration far into the surrounding exterior domain. The prescribed flux, J_0 , can include any arbitrary user-specified expressions. It can be a constant or a function of a dependent variable or independent variables.

UPSIDE INWARD FLUX

For 1D components, enter the **cross-sectional perimeter** P_c to get the out-of-plane flux

$$J_{0,i} = P_c J_{0,z,i}.$$

The default value of P_c is the circumference. Either keep the default value, for a circular cross-section shape, or edit the value to get a user-defined shape of the out-of-plane cross-section.

The available flux type options are **General inward flux** and **External convection**. Select the **Species** check box for the species for which to specify the flux, and enter a value or expression for the inward flux in the corresponding field. Use a minus sign when specifying a flux directed out of the system. To use another boundary condition for a specific species, click to clear the check box for that species.

Set **Flux type** to **External convection** to prescribe a flux to or from an exterior domain (not modeled) assumed to include convection. The exterior can for example include a forced convection to control the temperature or to increase the mass transport. In this case the prescribed mass flux corresponds to

$$J_0 = k_c(c_b - c)$$

where k_c is a mass transfer coefficient and c_b is the bulk concentration, the typical concentration far into the surrounding exterior domain.

DOWNSIDE INWARD FLUX

The available options are **General inward flux** and **External convection**. The settings are the same as for the [Upside Inward Flux](#) section.

FURTHER READING

For an example of using the Out-of-Plane Flux node, see this application example:



Mass Transfer from a Thin Domain: Application Library path
Chemical_Reaction_Engineering_Module/Tutorials/thin_domain

Fracture

Use this node to model mass transport along thin fractures in porous media. The node assumes that the transport in the tangential direction along the fracture is dominant, as a result of lower flow resistance.



This feature is only available in a limited set of add-on products. See <https://www.comsol.com/products/specifications/> for more details on availability.

FRACTURE PROPERTIES

Specify a value for the **Fracture thickness** d_f .

MATRIX PROPERTIES

Use the **Porous material** list to define a material specifying the matrix properties on the current selection. By default the **Boundary material** is used.

Specify the **Porosity**, ε_p (dimensionless) of the porous matrix. This is by default taken **From material**. Select **User defined** to instead enter a different value.

CONVECTION



Select an option from the **Velocity field** list to specify the convective velocity along the fracture. For a consistent model, use a Fracture Flow feature in a Darcy's Law interface to compute the fluid flow velocity in the fracture.

For **User defined**, enter values or expressions for the velocity components in the table shown.



The settings for the **Diffusion** is the similar to the settings for the diffusion coefficients in the **Fluid** node. The **Dispersion** settings are identical to the ones in the **Dispersion** node.

The Transport of Diluted Species in Fractures Interface

The **Transport of Diluted Species in Fractures (dsf)** interface () is found under the **Chemical Species Transport** branch () and is used to model the transport of a solute species along thin porous fractures, taking into account diffusion, dispersion, convection, and chemical reactions. The fractures are defined by boundaries and the solute species is assumed to be present in a solvent.

The interface supports simulation of species transport along boundaries in 2D and 3D, and axisymmetric components in 2D. The dependent variable is the molar concentration, c . Modeling multiple species transport is possible, whereby the physics interface solves for the molar concentration, c_i , of each species i .



This interface is only available in a limited set of add-on products. For a detailed overview of which interfaces are available in each product, visit <https://www.comsol.com/products/specifications/>

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 **dsf**.


BOUNDARY SELECTION

If model geometry includes boundaries that should not be included in the mass transfer simulation, remove those from the selection list.

TRANSPORT MECHANISMS


Mass transport due to diffusion is always included. Use the **Convection** check box, available under **Additional transport mechanisms**, to control whether to also include convective transport.

CONSISTENT STABILIZATION


To display this sections, click the **Show** button () and select **Stabilization**. Use this section to control the application of the available consistent stabilization methods; **Streamline diffusion** and **Crosswind diffusion**.

- When the **Crosswind diffusion** check box is selected, a weak term that reduces spurious oscillations is added to the transport equation. The resulting equation system is always nonlinear. There are two options for the **Crosswind diffusion type**:
 - **Do Carmo and Galeão** — the default option. This type of crosswind diffusion reduces undershoots and overshoots to a minimum but can in rare cases give equation systems that are difficult to fully converge.
 - **Codina**. This option is less diffusive compared to the Do Carmo and Galeão option but can result in more undershoots and overshoots. It is also less effective for anisotropic meshes. The Codina option activates a text field for the **Lower gradient limit** g_{lim} . It defaults to $0.1 [\text{mol}/\text{m}^3] / \text{tds.helem}$, where tds.helem is the local element size.
- For both consistent stabilization methods select an **Equation residual**. **Approximate residual** is the default and means that derivatives of the diffusion tensor components are neglected. This setting is usually accurate enough and is computationally faster. If required, select **Full residual** instead.

INCONSISTENT STABILIZATION

To display this section, click the **Show** button () and select **Stabilization**. By default, the **Isotropic diffusion** check box is not selected, because this type of stabilization adds artificial diffusion and affects the accuracy of the original problem. However, this option can be used to get a good initial guess for under resolved problems.

DISCRETIZATION



To display all settings available in this section, click the **Show** button () and select **Advanced Physics Options**.

The **Value type when using splitting of complex variables** setting should in most pure mass transfer problems be set to **Real**, which is the default. It makes sure that the dependent variable does not get affected by small imaginary contributions, which can occur, for example, when combining a Time Dependent or Stationary study with a frequency-domain study. For more information, see [Splitting Complex-Valued Variables](#) in the *COMSOL Multiphysics Reference Manual*.

DEPENDENT VARIABLES

The dependent variable name is **Concentration** *c* by default. A dependent variable name must be unique with respect to all other dependent variables in the component.

Add or remove species variables in the model and also change the names of the dependent variables that represent the species concentrations.

Enter the **Number of species**. Use the **Add concentration** () and **Remove concentration** () buttons as needed.

FURTHER READING



- [Mass Transport in Fractures](#) in the theory section.
- [Numerical Stabilization](#) in the *COMSOL Multiphysics Reference Manual*.
- [Domain, Boundary, and Pair Nodes for the Transport of Diluted Species Interface](#)
- In the *COMSOL Multiphysics Reference Manual*, see [Table 2-4](#) for links to common sections and [Table 2-5](#) for common feature nodes. You can also search for information: press F1 to open the **Help** window or Ctrl+F1 to open the **Documentation** window.

Boundary, Edge, Point, and Pair Nodes for the Transport of Diluted Species in Fractures Interface

The [Transport of Diluted Species in Fractures Interface](#) has the following boundary, edge, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or by right-clicking to access the context menu (all users).

- | | |
|---------------------------------|----------------------------------|
| • Adsorption | • Initial Values |
| • Concentration | • No Flux |
| • Flux | • Outflow |
| • Fracture | • Reactions |
| • Inflow | • Species Source |

Adsorption

Use this node to model adsorption of the fluid phase species onto the porous media surface of the fracture.

MATRIX PROPERTIES

Use the **Porous material** list to define a material specifying the matrix properties on the current selection. By default the **Domain material** is used. The density of the porous media is needed when modeling adsorption to the surface of the porous matrix. By default **Density** ρ is set to from domain material.

ADSORPTION

Select an **Adsorption isotherm** — **Langmuir** (the default), **Freundlich**, **Toth**, **BET**, or **User defined** to specify how to compute c_P , the amount of species sorbed to the solid phase (moles per unit dry weight of the solid):

- For **Langmuir**:

$$c_P = c_{P\max} \frac{K_L c}{1 + K_L c} \quad \frac{\partial c_P}{\partial c} = \frac{K_L c_{P\max}}{(1 + K_L c)^2}$$

Enter a **Langmuir constant** $k_{L,c}$ (SI unit: m^3/mol) and an **Adsorption maximum** $c_{P,\max,c}$ (SI unit: mol/kg):

- For **Freundlich**:

$$c_P = K_F \left(\frac{c}{c_{\text{ref}}} \right)^N \quad \frac{\partial c_P}{\partial c} = N \frac{c_P}{c} \quad \text{Freundlich}$$

Enter a **Freundlich constant** $k_{F,c}$ (SI unit: mol/kg), a **Freundlich exponent** $N_{F,c}$ (dimensionless), and a **Reference concentration** $c_{\text{ref},c}$ (SI unit: mol/m^3).

- For **Toth**:

$$c_P = c_{P\max} \frac{b_T c}{(1 + (b_T c)^{N_T})^{1/N_T}}$$

and

$$\frac{\partial c_p}{\partial c} = c_{p,\max} b_T (1 + (b_T c)^{N_{T,c}})^{-\left(1 + \frac{1}{N_{T,c}}\right)}$$

Enter a **Toth constant** $b_{T,c}$ (SI unit: m^3/mol), a **Toth exponent** $N_{T,c}$ (dimensionless), and an **Adsorption maximum** $c_{p,\max,c}$ (SI unit: mol/kg).

- For **BET (Brunauer-Emmett-Teller)**:

$$c_p = \frac{K_B c_0 c}{(c_S - c) \left(1 + (K_B - 1) \frac{c}{c_S} \right)}$$

and

$$\frac{K_B c_0 c_S ((K_B - 1) c^2 + c_S^2)}{(c - c_S)^2 (c_S + (K_B - 1) c)^2}$$

Enter a **BET constant** $K_{B,c}$ (dimensionless), a **Monolayer adsorption capacity** $c_{0,c}$ (SI unit: mol/kg), and an **Saturation concentration** $c_{S,c}$ (SI unit: mol/m^3).

- For **User defined**:

$$c_p = f(c)$$

Enter an adsorption isotherm $c_{p,c}$ (SI unit: mol/kg).

For more information, see [Adsorption](#) in the theory section.

FURTHER READING

See the theory chapter in the section [Mass Balance Equation for Transport of Diluted Species in Porous Media](#).

Concentration


Use this node to specify the species concentration on a fracture boundary (applied in points in 2D and along edges in 3D). For example, a $c = c_0$ condition specifies the concentration of species c .

CONCENTRATION

Individually specify the concentration for each species. Select the check box for the **Species** to specify the concentration, and then enter a value or expression in the

corresponding field. To use another boundary condition for a specific species, click to clear the check box for the concentration of that species.

CONSTRAINT SETTINGS

To display this section, click the **Show** button () and select **Advanced Physics Options**. You can find details about the different constraint settings in the section [Constraint Reaction Terms](#) in the *COMSOL Multiphysics Reference Manual*.

Flux

This node can be used to specify the species flux across a boundary of a porous fracture (applied in points in 2D and along edges in 3D). The flux of species c is defined as

$$\mathbf{n} \cdot (D_c \nabla c) = N_0$$

where N_0 is an arbitrary user-specified flux expression. For example, N_0 can represent a flux due to chemical reactions, or a phase change. A positive N_0 implies that the concentration inside the fracture increases.

INWARD FLUX

Specify the flux of each species individually. To use another boundary condition for a specific species, click to clear the check box for the mass fraction of that species.

Fracture

Use this node to model mass transport along thin fracture surfaces situated inside porous or solid material. The node assumes that the transport in the tangential direction of the fracture is dominant, as a result of lower flow resistance. Note that the fracture it self is modeled as porous.

MATRIX PROPERTIES

Use the **Porous material** list to define a material specifying the matrix properties on the current selection. By default the **Boundary material** is used.

Specify the **Porosity**, ε_p (dimensionless) of the porous matrix. This is by default taken **From material**. Select **User defined** to instead enter a different value.

CONVECTION

Select an option from the **Velocity field** list to specify the convective velocity along the fracture. For a consistent model, use [The Fracture Flow Interface](#) to compute the fluid flow velocity.

For **User defined**, enter values or expressions for the velocity components in the table shown.



The settings for the **Diffusion** is the similar to the settings for the diffusion coefficients in the **Fluid** node. The **Dispersion** settings are identical to the ones in the **Dispersion** node.

Inflow

Use this node to specify all species concentrations at a fracture inlet. The condition is applied in points in 2D and along edges in 3D.

If you want to specify the concentration of a subset of the partaking species, this can be done by using the [Concentration](#) node instead.

CONCENTRATION


For the concentration of each species $c_{0,c}$ (SI unit: mol/m³), enter a value or expression.

BOUNDARY CONDITION TYPE

This section in the settings is only available for some products. Search for “Inflow” on the page: <https://www.comsol.com/products/specifications/> for more details on availability.

The option **Concentration constraint** constrains the concentration values on the boundary by the use of pointwise constraints. The other option, **Flux (Danckwerts)** can be more stable and fast to solve when high reaction rates are anticipated in the vicinity of the inlet. Oscillations on the solutions can also be avoided in such cases. The latter condition uses a flux boundary condition based on the velocity across the boundary and the concentration values. See further details in the theory section.

CONSTRAINT SETTINGS

To display this section, click the **Show** button () and select **Advanced Physics Options**. You can find details about the different constraint settings in the section [Constraint Reaction Terms](#) in the *COMSOL Multiphysics Reference Manual*.

FURTHER READING

See the theory chapter in the section [Danckwerts Inflow Boundary Condition](#).

No Flux

This node can be used to specify that the species flux across a boundary of a porous fracture is zero. The condition is applied in points in 2D and along edges in 3D.

Outflow

Set this condition at fracture outlets where species are transported out of the model domain by fluid motion. The condition is applied in points in 2D and along edges in 3D. It is assumed that convection is the dominating transport mechanism across outflow boundaries, and therefore that diffusive transport can be ignored, that is:

$$\mathbf{n} \cdot (-D_c \nabla c) = 0$$

Reactions

Use the **Reactions** node to account for the consumption or production of species through chemical reactions in the fracture. Define the rate expressions as required.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries on which to define rate expression or expressions that govern the source term in the transport equations.

Several reaction nodes can be used to account for different reactions in different parts of the fracture.

REACTION RATES

Add a rate expression R_i for species i . Enter a value or expression in the field. Note that if you have the Chemistry interface available, included with the Chemical Reaction Engineering Module, the reaction rate expressions can be automatically generated and picked up using the drop-down menu.

REACTING VOLUME

When specifying reaction rates for a species in a fracture, the specified reaction rate may have the basis of the pore volume of the fracture, or the total volume.

- For **Total volume**, the reaction expressions in are specified per unit volume of the fracture. The reaction expressions will be multiplied by the fracture thickness d_{fr} .
- For **Pore volume**, the reaction expressions in $\text{mol}/(\text{m}^3 \cdot \text{s})$ are specified per unit volume of total pore space in the fracture. The reaction expressions will be multiplied by the fracture thickness d_{fr} and the fracture porosity, ϵ_p .

Species Source

In order to account for consumption or production of species in a fracture, the **Species Source** node adds source terms expressions S_i to the right-hand side of the species transport equations.

BOUNDARY SELECTION

From the **Selection** list, choose the boundaries on which to define expressions that govern the source term in the transport equations.

If there are several different parts of the fracture, with subsequent and different sources occurring within them, it might be necessary to remove some boundaries from the selection. The sources in these can then be defined using an additional **Species Source** node.

SPECIES SOURCE

Add a source term S_i for each of the species solved for. Enter a value or expression in the field of the corresponding species.

The Moisture Transport Interfaces

For the Porous Media Flow Module there are two Moisture Transport Interfaces available under the **Chemical Species Transport** branch: The Moisture Transport in Building Materials Interface and the Moisture Transport in Air Interface. They are used to model moisture transport in building materials (by taking into account moisture storage, liquid transport by capillary suction forces) and in moist air (by convection and diffusion of vapor in moist air), respectively.

Both interfaces have been implemented in the Heat Transfer Module and made available for the Porous Media Flow Module. Therefore, they are described in the *Heat Transfer Module User's Guide*, specifically in the following chapters:

- [The Moisture Transport in Porous Media Interface](#)
- [The Moisture Transport in Building Materials Interface](#)
- [The Moisture Transport Features](#)
- [Theory for Moisture Transport](#)

Theory for the Transport of Diluted Species Interface

The [Transport of Diluted Species Interface](#) provides a predefined modeling environment for studying the evolution of chemical species transported by diffusion and convection. The physics interface assumes that all species present are dilute; that is, that their concentration is small compared to a solvent fluid or solid. As a rule of thumb, a mixture containing several species can be considered dilute when the concentration of the solvent is more than 90 mol%. Due to the dilution, mixture properties such as density and viscosity can be assumed to correspond to those of the solvent.

When studying mixtures that are not dilute, the mixture and transport properties depend on the composition, and a different physics interface is recommended. See [The Transport of Concentrated Species Interface](#) in the *Chemical Reaction Engineering Module User's Guide* for more information.

Fick's law governs the diffusion of the solutes, dilute mixtures, or solutions, while the phenomenon of ionic migration is sometimes referred to as *electrokinetic flow*. The Transport of Diluted Species interface supports the simulations of chemical species transport by convection, migration, and diffusion in 1D, 2D, and 3D as well as for axisymmetric components in 1D and 2D.

In this section:

- [Adding Transport Through Migration](#)
- [Convective Term Formulation](#)
- [Crosswind Diffusion](#)
- [Danckwerts Inflow Boundary Condition](#)
- [Equilibrium Reaction Theory](#)
- [Mass Balance Equation](#)
- [Mass Sources for Species Transport](#)
- [Solving a Diffusion Equation Only](#)
- [Supporting Electrolytes](#)
- [References](#)

Note: Some features explained in this section require certain add-on modules. For details see <https://www.comsol.com/products/specifications/>

The section also includes the theory for [The Transport of Diluted Species in Porous Media Interface](#):

- [Adsorption](#)
- [Convection in Porous Media](#)
- [Diffusion in Porous Media](#)
- [Dispersion](#)
- [Mass Balance Equation for Transport of Diluted Species in Porous Media](#)
- [Mass Transport in Fractures](#)
- [Reactions](#)

Mass Balance Equation

The default node attributed to the Transport of Diluted Species interface models chemical species transport through diffusion and convection and solves the mass conservation equation for one or more chemical species i :

$$\frac{\partial c_i}{\partial t} + \nabla \cdot \mathbf{J}_i + \mathbf{u} \cdot \nabla c_i = R_i \quad (5-2)$$

[Equation 5-2](#) in its form above includes the transport mechanisms diffusion and convection. If *Migration in Electric Field* is activated (only available in some add-on products), the migration mechanism will be added to the equation as well. See more details in the section [Adding Transport Through Migration](#).

- c_i is the concentration of the species (SI unit: mol/m³)
- D_i denotes the diffusion coefficient (SI unit: m²/s)
- R_i is a reaction rate expression for the species (SI unit: mol/(m³·s))
- \mathbf{u} is the mass averaged velocity vector (SI unit: m/s)
- \mathbf{J}_i is the mass flux diffusive flux vector (SI unit: mol/(m²·s))

The mass flux relative to the mass averaged velocity, \mathbf{J}_i (SI unit: mol/(m²·s)), is associated with the mass balance equation above and used in boundary conditions and flux computations. The Transport of Diluted Species interface always includes mass transport due to molecular diffusion. In this case the mass flux \mathbf{J}_i defines the diffusive flux vector

$$\mathbf{J}_i = -D_i \nabla c_i \quad (5-3)$$

An input field for the diffusion coefficient is available.

When *Migration in Electric Fields* is activated, the migration term is also added to the diffusive flux vector as shown in the section [Adding Transport Through Migration](#).

The third term on the left side of [Equation 5-2](#) describes the convective transport due to a velocity field \mathbf{u} . This field can be expressed analytically or obtained from coupling the physics interface to one that solves for fluid flow, such as *Laminar Flow*. Note that all fluid flow interfaces solve for the mass averaged velocity.

On the right-hand side of the mass balance equation ([Equation 5-2](#)), R_i represents a source or sink term, typically due to a chemical reaction or desorption on a porous matrix. To specify R_i , another node must be added to the Transport of Diluted Species interface — the **Reaction** node for example, which includes an input field for specifying a reaction expression using the variable names of all participating species.

Equilibrium Reaction Theory

The feature Equilibrium Reaction is described in this section. A chemical equilibrium reaction system is defined by the stoichiometry of the reaction and the relation between the chemical activities of the chemical species participating in the reaction (the equilibrium condition).

The kinetics of the reaction is so fast that the equilibrium condition is fulfilled at all times in all space coordinates.

The equilibrium condition is commonly based on the stoichiometric coefficients, ν_i (dimensionless), of the reaction; the species activities of the reacting species a_i (dimensionless); and an equilibrium constant, K_{eq} (1) according to:

$$K_{\text{eq}} = \frac{\prod_{i \in \text{products}} a_i^{\nu_i}}{\prod_{i \in \text{reactants}} a_i^{-\nu_i}}$$

where the species activities are defined as

$$a_i = \gamma_{c,i} \frac{c_i}{c_{a0}}$$

where c_{a0} (SI unit: mol/m³) is the standard molarity, and $\gamma_{c,i}$ (dimensionless) an activity coefficient.

Defining the stoichiometric coefficients positive for products and negative for reactants, the above equilibrium condition can also be written:

$$K_{\text{eq}} = \prod_i a_i^{v_i}$$

The **Equilibrium Reaction** node solves for a reaction rate so that the equilibrium condition is always fulfilled in the domain. It is available for the Chemical Reaction Engineering Module; Battery Design Module; Corrosion Module; Electrochemistry Module; Electrodeposition Module; and Fuel Cell & Electrolyzer Module.



$\gamma_{e,i}$ is set to unity when the Equilibrium constant is selected on the Settings window. For nonunity activity coefficients, a user defined equilibrium condition can be used.

EQUILIBRIUM REACTIONS AND INFLOW BOUNDARY CONDITIONS

Contradictory constraints arise if the boundary conditions for concentrations or activities are set so that the domain equilibrium condition is not fulfilled. Special treatment is therefore needed at Inflow boundaries, where the concentrations are set for all species in the mass transport interfaces.

One way of avoiding competing constraints on an inflow boundary is to add an additional reaction coordinate degree of freedom, solved for to create a set of modified inflow concentrations that fulfill the domain equilibrium condition. The reaction coordinate gives rise to a concentration shift, which is the offset to the inflow concentrations provided by the user. The shift for each species obeys the stoichiometry of the reaction and the equilibrium expression. The modified inflow concentrations are then used in the boundary conditions for the domain mass transport equations. The resulting modified inflow concentrations can be seen as the stationary solution for a batch reactor with the user inflow concentrations as initial concentrations. In addition, the domain reaction rate degree of freedom of the equilibrium reaction is constrained to zero on all Inflow boundaries.

EQUILIBRIUM REACTIONS AND CONCENTRATION BOUNDARY CONDITIONS

No special treatment is made with regards to input concentration values of the Concentration boundary node. Using this feature, you can explicitly set one or a set of concentrations, and the equilibrium condition acts on the rest of the concentrations.

However, there is no solution to the problem if more concentrations than the number of species minus the number of equilibrium reactions are set using this feature.

EQUILIBRIUM REACTIONS AND TIME-DEPENDENT SIMULATIONS

Spurious oscillations may occur in a time-dependent problem if the initial conditions do not fulfill the equilibrium condition. Since equilibrium reactions are assumed to be infinitely fast, the solution is to initialize the problem using an additional study step, solving for a stationary problem with all nonequilibrium reaction rates set to zero. Manual scaling of the reaction rate dependent variables is needed in this study step.

Convective Term Formulation


The default node attributed to [The Transport of Diluted Species Interface](#) assumes chemical species transport through diffusion and convection (depending on the modules licensed, a check box to activate migration is available) and implements the mass balance equation in [Equation 5-2](#).

There are two ways to present a mass balance where chemical species transport occurs through diffusion and convection. These are the nonconservative and conservative formulations of the convective term:

$$\text{nonconservative: } \frac{\partial c}{\partial t} + \mathbf{u} \cdot \nabla c = \nabla \cdot \mathbf{J}_i + R \quad (5-4)$$

$$\text{conservative: } \frac{\partial c}{\partial t} + \nabla \cdot (c\mathbf{u}) = \nabla \cdot \mathbf{J}_i + R \quad (5-5)$$

and each is treated slightly differently by the solver algorithms. In these equations \mathbf{J}_i (SI unit: $\text{mol}/(\text{m}^2 \cdot \text{s})$) is the diffusive flux vector, R (SI unit: $\text{mol}/(\text{m}^3 \cdot \text{s})$) is a production or consumption rate expression, and \mathbf{u} (SI unit: m/s) is the solvent velocity field. The diffusion process can be anisotropic, in which case D is a tensor.

If the conservative formulation is expanded using the chain rule, then one of the terms from the convection part, $c\nabla \cdot \mathbf{u}$, would equal zero for an incompressible fluid and would result in the nonconservative formulation above. This is in fact the default formulation in this physics interface. To switch between the two formulations, click the **Show** button () and select **Advanced Physics Options**.

Solving a Diffusion Equation Only

Remove the convection term from Equation 5-4 and Equation 5-5 by clearing the Convection check box in the Transport Mechanisms section for The Transport of Diluted Species Interface. The equation then becomes

$$\frac{\partial c}{\partial t} = \nabla \cdot \mathbf{J}_i + R$$

Mass Sources for Species Transport

Note: The features below are only available in a limited set of add-on products. For a detailed overview of which features are available in each product, visit <https://www.comsol.com/products/specifications/>

There are two types of mass sources in the Transport of Diluted Species interface: point sources and line sources.

POINT SOURCE

A point source is theoretically formed by assuming a mass injection/ejection, \dot{Q}_c (SI unit: mol/(m³·s)), in a small volume δV and then letting the size of the volume tend to zero while keeping the total mass flux constant. Given a point source strength, $\dot{q}_{p,c}$ (SI unit: mol/s), this can be expressed as

$$\lim_{\delta V \rightarrow 0} \int_{\delta V} \dot{Q}_c = \dot{q}_{p,c} \quad (5-6)$$

An alternative way to form a point source is to assume that mass is injected/extracted through the surface of a small object. Letting the object surface area tend to zero while keeping the mass flux constant results in the same point source. For this alternative approach, effects resulting from the physical object's volume need to be neglected.

The weak contribution

$$\dot{q}_{p,c} \text{test}(c)$$

is added at a point in the geometry. As can be seen from Equation 5-6, \dot{Q}_c must tend to plus or minus infinity as δV tends to zero. This means that in theory the concentration also tends to plus or minus infinity.

Observe that “point” refers to the physical representation of the source. A point source can therefore only be added to points in 3D components and to points on the symmetry axis in 2D axisymmetry components. Other geometrical points in 2D components represent physical lines.

The finite element representation of Equation 5-6 corresponds to a finite concentration at a point with the effect of the point source spread out over a region around the point. The size of the region depends on the mesh and on the strength of the source. A finer mesh gives a smaller affected region but also a more extreme concentration value. It is important not to mesh too finely around a point source since this can result in unphysical concentration values. It can also have a negative effect on the condition number for the equation system.

LINE SOURCE

A line source can theoretically be formed by assuming a source of strength $\dot{Q}_{l,c}$ (SI unit: mol/(m³·s)), located within a tube with cross section δS and then letting δS tend to zero while keeping the total mass flux per unit length constant. Given a line source strength, $\dot{q}_{l,c}$ (SI unit: mol/(m·s)), this can be expressed as

$$\lim_{\delta S \rightarrow 0} \int_{\delta S} \dot{Q}_{l,c} = \dot{q}_{l,c} \quad (5-7)$$

As in the point source case, an alternative approach is to assume that mass is injected/extracted through the surface of a small object. This results in the same mass source, but requires that effects resulting from the physical object’s volume are neglected.

The weak contribution

$$\dot{q}_{l,c} \text{test}(c)$$

is added on lines in 3D or at points in 2D (which represent cut-through views of lines). Line sources can also be added on the axisymmetry line in 2D axisymmetry components. It cannot, however, be added on geometrical lines in 2D since those represent physical planes.

As with a point source, it is important not to mesh too finely around the line source.



For feature node information, see [Line Mass Source](#) and [Point Mass Source](#).

Note: Migration is only available in a limited set of add-on products. For a detailed overview of which features are available in each product, visit <https://www.comsol.com/products/specifications/>

In addition to transport due to convection and diffusion, the Transport of Diluted Species interface supports ionic species transport by *migration*. This is done by selecting the Migration in Electric Field check box under the Transport Mechanisms section for the physics interface. The mass balance then becomes:

$$\frac{\partial c_i}{\partial t} + \nabla \cdot (-D_i \nabla c_i - z_i u_{m,i} F c_i \nabla V + c_i \mathbf{u}) = R_i \quad (5-8)$$

where

- c_i (SI unit: mol/m³) denotes the concentration of species i
- D_i (SI unit: m²/s) is the diffusion coefficient of species i
- \mathbf{u} (SI unit: m/s) is the fluid velocity
- F (SI unit: A·s/mol) refers to Faraday's constant
- V (SI unit: V) denotes the electric potential
- z_i (dimensionless) is the charge number of the ionic species, and
- $u_{m,i}$ (SI unit: mol·s/kg) is its ionic mobility

In this case the diffusive flux vector is

$$\mathbf{J}_i = -D_i \nabla c_i - z_i u_{m,i} F c_i \nabla V$$

The velocity, \mathbf{u} , can be a computed fluid velocity field from a Fluid Flow interface or a specified function of the spatial variables x , y , and z . The potential can be provided by an expression or by coupling the system of equations to a current balance, such as the Electrostatics interface. Sometimes it is assumed to be a supporting electrolyte present, which simplifies the transport equations. In that case, the modeled charged species concentration is very low compared to other ions dissolved in the solution. Thus, the species concentration does not influence the solution's conductivity and the net charge within the fluid.

The Nernst–Einstein relation can in many cases be used for relating the species mobility to the species diffusivity according to

$$u_{m,i} = \frac{D_i}{RT}$$

where R (SI unit: J/(mol·K)) is the molar gas constant and T (SI unit: K) is the temperature.

Note: In the *Nernst–Planck Equations* interface, the ionic species contribute to the charge transfer in the solution. It includes an *electroneutrality condition* and also computes the electric potential field in the electrolyte. For more information, see [Theory for the Nernst–Planck Equations Interface](#). This interface is included in the Chemical Reaction Engineering Module.

Supporting Electrolytes

In *electrolyte* solutions, a salt can be added to provide a high electrolyte conductivity and decrease the ohmic losses in a cell. These solutions are often called *supporting electrolytes*, buffer solutions, or carrier electrolytes. The added species, a negative and a positive ion pair, predominates over all other species. Therefore, the supporting electrolyte species can be assumed to dominate the current transport in the solution. In addition, the predominant supporting ions are usually selected so that they do not react at the electrode surfaces since the high conductivity should be kept through the process, that is, they should not be electro-active species. This also means that the concentration gradients of the predominant species in a supporting electrolyte are usually negligible.

Modeling and solving for a supporting electrolyte in the *Electrostatics* or *Secondary Current Distribution* interfaces will give a potential distribution that drives the migration in the Transport of Diluted Species Interface.

The current density vector is proportional to the sum of all species fluxes as expressed by Faraday’s law:

$$\mathbf{i} = F \sum_i z_i \mathbf{N}_i$$

The electroneutrality condition ensures that there is always a zero net charge at any position in a dilute solution. Intuitively, this means that it is impossible to create a current by manually pumping positive ions in one direction and negative ions in the other. Therefore, the convective term is canceled out to yield the following expression for the electrolyte current density, where j denotes the supporting species:

$$\mathbf{i} = F \sum_j -z_j^2 u_{m,j} F c_j \nabla \phi \quad (5-9)$$

Equation 5-9 is simply Ohm's law for ionic current transport and can be simplified to

$$\mathbf{i} = -\kappa \nabla \phi \quad (5-10)$$

where κ is the conductivity of the supporting electrolyte. A current balance gives the current and potential density in the cell

$$\nabla \cdot \mathbf{i} = 0$$

which, in combination with Equation 5-10, yields:

$$\nabla \cdot (-\kappa \nabla \phi) = 0 \quad (5-11)$$

Equation 5-11 can be easily solved using the Electrostatics or Secondary Current Distribution interface and, when coupled to the Transport in Diluted Species interface, the potential distribution shows up in the migration term.

Crosswind Diffusion

Transport of diluted species applications can often result in models with a very high cell Péclet number — that is, systems where convection or migration dominates over diffusion. Streamline diffusion and crosswind diffusion are of paramount importance to obtain physically reasonable results. The Transport of Diluted Species interface provides two crosswind diffusion options using different formulations. Observe that crosswind diffusion makes the equation system nonlinear even if the transport equation is linear.

DO CARMO AND GALEÃO

This is the formulation described in [Numerical Stabilization](#) in the *COMSOL Multiphysics Reference Manual*. The method reduces over- and undershoots to a minimum, even for anisotropic meshes.

In some cases, the resulting nonlinear equation system can be difficult to converge. This can happen when the cell Péclet number is very high and the model contains many thin layers, such as contact discontinuities. You then have three options:

- Refine the mesh, especially in regions with thin layers.
- Use a nonlinear solver with a constant damping factor less than one.
- Switch to the Codina crosswind formulation.

CODINA

The Codina formulation is described in [Ref. 1](#). It adds diffusion strictly in the direction orthogonal to the streamline direction. Compared to the do Carmo and Galeão formulation, the Codina formulation adds less diffusion but is not as efficient at reducing over- and undershoots. It also does not work as well for anisotropic meshes. The advantage is that the resulting nonlinear system is easier to converge and that underresolved gradients are less smeared out.

Danckwerts Inflow Boundary Condition

Constraining the composition to fixed values at an inlet to a reactor may sometimes result in issues with unreasonably high reaction rates or singularities at the inlet boundary. These problems may many times be mitigated by using a flux boundary condition instead, based on the upstream concentrations and the fluid velocity at the boundary. In chemical engineering, this type of flux boundary condition is also known as a Danckwerts condition.

Use the Danckwerts condition to specify inlet conditions for domains where high reaction rates are anticipated in the vicinity to the inlet ([Ref. 2](#)).

Given an upstream concentration $c_{i,0}$, the Danckwerts inflow boundary condition prescribed the total flux as

$$\mathbf{n} \cdot (\mathbf{J}_i + \mathbf{u}c_i) = \mathbf{n} \cdot (\mathbf{u}c_{i,0}) \quad (5-12)$$

VARIABLY SATURATED POROUS MEDIA

The following equations for the molar concentrations, c_i , describe the transport of solutes in a variably saturated porous medium for the most general case, when the pore space is primarily filled with liquid but also contain pockets or immobile gas:

$$\frac{\partial}{\partial t}(\theta_l c_i) + \frac{\partial}{\partial t}(\rho c_{P,i}) + \frac{\partial}{\partial t}(\theta_g c_{G,i}) + \mathbf{u} \cdot \nabla c_i = \nabla \cdot [(D_{D,i} + D_{e,i}) \nabla c_i] + R_i + S_i \quad (5-13)$$

On the left-hand side of Equation 5-13, the first three terms correspond to the accumulation of species within the liquid, solid, and gas phases, while the last term describes the convection due to the velocity field \mathbf{u} (SI unit: m/s).

In Equation 5-13 c_i denotes the concentration of species i in the liquid (SI unit: mol/m³), $c_{P,i}$ the amount adsorbed to solid particles (moles per unit dry weight of the solid), and $c_{G,i}$ the concentration of species i in the gas phase.

The equation balances the mass transport throughout the porous medium using the porosity ϵ_p , the liquid volume fraction θ_l ; the dry bulk density, $\rho = (1 - \epsilon_s)\rho_s$, and the solid phase density ρ_s .

For saturated porous media, the liquid volume fraction θ_l is equal to the porosity ϵ_p , but for unsaturated porous media, they are related by the saturation s as $\theta_l = \epsilon_p s$. The resulting gas volume fraction in the case of an unsaturated porous medium is

$$\theta_g = \epsilon_p - \theta_l = (1 - s)\epsilon_p$$

On the right-hand side of Equation 5-13, the first term introduces the spreading of species due to mechanical mixing resulting from the porous media (dispersion), as well as from diffusion and volatilization to the gas phase. The dispersion tensor is denoted D_D (SI unit: m²/s) and the effective diffusion by D_e (SI unit: m²/s).

The last two terms on the right-hand side of Equation 5-13 describe production or consumption of the species; R_i is a reaction rate expression which can account for reactions in the liquid, solid, or gas phase, and S_i is an arbitrary source term, for example due to a fluid flow source or sink.

Adsorption

The time evolution of the adsorption, the solute transport to or from the solid phase, is defined by assuming that the amount of solute adsorbed to the solid, $c_{p,i}$, is a function of the concentration in the fluid c_i . This implies that the solute concentration in the liquid and solid phase are in instant equilibrium. The adsorption term can be expanded to give

$$\frac{\partial}{\partial t}(\rho c_{p,i}) = \rho \frac{\partial c_{p,i}}{\partial c_i} \frac{\partial c_i}{\partial t} - c_{p,i} \rho_s \frac{\partial \epsilon_p}{\partial t} = \rho K_{p,i} \frac{\partial c_i}{\partial t} - c_{p,i} \rho_s \frac{\partial \epsilon_p}{\partial t} \quad (5-14)$$

where $K_{p,i} = \partial c_{p,i} / \partial c_i$ is the adsorption isotherm.

Volatilization

Volatilization is the process where a solute species in the liquid is transported to the gas phase due to vaporization. Assuming that the amount of solute in the gas phase, $c_{G,i}$, is a linear function of the liquid phase concentration, the volatilization term is defined as

$$\frac{\partial}{\partial t} \theta_g c_{G,i} = \theta_g \frac{\partial c_{G,i}}{\partial c_i} \frac{\partial c_i}{\partial t} + k_{G,i} c_i \frac{\partial \theta_g}{\partial t} = \theta_g k_{G,i} \frac{\partial c_i}{\partial t} + k_{G,i} c_i \frac{\partial \theta_g}{\partial t} \quad (5-15)$$

where $k_{G,i} = \partial c_{G,i} / \partial c_i$ is the linear volatilization.

SATURATED POROUS MEDIA

In the case of transport in a saturated porous medium, $\theta_l = \epsilon_p$, and the governing equations are

$$\frac{\partial}{\partial t}(\epsilon_p c_i) + \frac{\partial}{\partial t}(\rho c_{p,i}) + \mathbf{u} \cdot \nabla c_i = \nabla \cdot [(D_{D,i} + D_{e,i}) \nabla c_i] + R_i + S_i \quad (5-16)$$

Convection in Porous Media

Convection (also called advection) describes the movement of a species, such as a pollutant, with the bulk fluid velocity. The velocity field \mathbf{u} corresponds to a superficial volume average over a unit volume of the porous medium, including both pores and matrix. This velocity is sometimes called *Darcy velocity*, and defined as volume flow

rates per unit cross section of the medium. This definition makes the velocity field continuous across the boundaries between porous regions and regions with free flow.



The velocity field to be used in the Model Inputs section on the physics interface can, for example, be prescribed using the velocity field from a Darcy's Law or a Brinkman Equations interface.

The average linear fluid velocities \mathbf{u}_a , provides an estimate of the fluid velocity within the pores:

$$\mathbf{u}_a = \frac{\mathbf{u}}{\varepsilon_p} \quad \text{Saturated}$$

$$\mathbf{u}_a = \frac{\mathbf{u}}{\theta_l} \quad \text{Unsaturated}$$

where ε_p is the porosity and $\theta_l = s\varepsilon_p$ the liquid volume fraction, and s the saturation, a dimensionless number between 0 and 1.

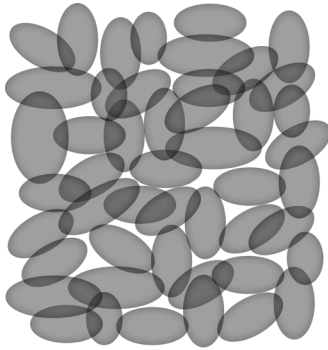


Figure 5-2: A block of a porous medium consisting of solids and the pore space between the solid grains. The average linear velocity describes how fast the fluid moves within the pores. The Darcy velocity attributes this flow over the entire fluid-solid face.

CONVECTIVE TERM FORMULATION


The Transport of Diluted Species in Porous Media interface includes two formulations of the convective term. The conservative formulation of the species equations in [Equation 5-13](#) is written as:

$$\frac{\partial}{\partial t}(\theta_l c_i) + \frac{\partial}{\partial t}(\rho c_{p,i}) + \frac{\partial}{\partial t}(\theta_g c_{g,i}) + \nabla \cdot \mathbf{u} c_i = \nabla \cdot [(D_{D,i} + D_{e,i}) \nabla c_i] + R_i + S_i \quad (5-17)$$

If the conservative formulation is expanded using the chain rule, then one of the terms from the convection part, $c_i \nabla \cdot \mathbf{u}$, would equal zero for an incompressible fluid and would result in the nonconservative formulation described in [Equation 5-13](#).

When using the nonconservative formulation, which is the default, the fluid is assumed incompressible and divergence free: $\nabla \cdot \mathbf{u} = 0$. The nonconservative formulation improves the stability of systems coupled to a momentum equation (fluid flow equation).



To switch between the two formulations, click the **Show** button () and select **Advanced Physics Options**. In the section **Advanced Settings** select either **Nonconservative form** (the default) or **Conservative form**. The conservative formulation should be used for compressible flow.

Diffusion in Porous Media

The effective diffusion in porous media, D_e , depends on the structure of the porous material and the phases involved. Dependent on whether the transport of diluted species occurs in free flow, saturated or unsaturated porous media, the effective diffusivity is defined as:

$$\begin{aligned} D_e &= D_F && \text{Free Flow} \\ D_e &= \frac{\epsilon_p}{\tau_L} D_L && \text{Saturated Porous Media} \\ D_e &= \frac{\theta_l}{\tau_L} D_L && \text{Unsaturated Porous Media} \\ D_e &= \frac{\theta_l}{\tau_L} D_L + \frac{\theta_g}{\tau_G} k_G D_G && \text{Unsaturated with Volatilization} \end{aligned}$$

Here D_F , D_L , and D_G are the single-phase diffusion coefficients for the species diluted in a fluid, a pure liquid, and a gas phase, respectively (SI unit: m^2/s), and τ_F , τ_L , and τ_G are the corresponding tortuosity factors (dimensionless).

The tortuosity factor accounts for the reduced diffusivity due to the fact that the solid grains impede Brownian motion. The interface provides predefined expressions to

compute the tortuosity factors in partially saturated porous media according to the Millington and Quirk model (Ref. 12):

$$\tau_L = \theta_l^{-7/3} \epsilon_p^2, \tau_G = \theta_g^{-7/3} \epsilon_p^2$$

and Bruggeman model

$$\tau_L = \theta_l^{-5/2} \epsilon_p^2, \tau_G = \theta_g^{-5/2} \epsilon_p^2$$

For saturated porous media $\theta_l = \epsilon_p$. The fluid tortuosity for the Millington and Quirk model is

$$\tau_L = \epsilon_p^{-1/3}$$

and for the Bruggeman model the tortuosity is defined as

$$\tau_L = \epsilon_p^{-1/2}$$

User defined expressions for the tortuosity factor can also be applied.

Dispersion

The contribution of dispersion to the mixing of species typically overshadows the contribution from molecular diffusion, except when the fluid velocity is very small.

The spreading of mass, as a fluid travel through a porous medium is caused by several contributing effects. Local variations in fluid velocity lead to mechanical mixing referred to as dispersion occurs because the fluid in the pore space flows around solid particles, so the velocity field varies within pore channels. The spreading in the direction parallel to the flow, or *longitudinal dispersivity*, typically exceeds the *transverse dispersivity* from up to an order of magnitude. Being driven by the concentration gradient alone, molecular diffusion is small relative to the mechanical dispersion, except at very low fluid velocities.

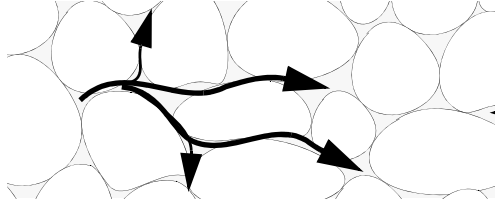


Figure 5-3: Spreading of fluid around solid particles in a porous medium.

The dispersion is controlled through the dispersion tensor D_D . The tensor components can either be given by user-defined values or expressions or derived from the directional dispersivities.

Using the longitudinal and transverse dispersivities in 2D, the dispersivity tensor components are ([Ref. 9](#)):

$$D_{Dii} = \alpha_L \frac{u_i^2}{|\mathbf{u}|} + \alpha_T \frac{u_j^2}{|\mathbf{u}|}$$

$$D_{Dij} = D_{Dji} = (\alpha_L - \alpha_T) \frac{u_i u_j}{|\mathbf{u}|}$$

In these equations, D_{Dii} (SI unit: m^2/s) are the principal components of the dispersivity tensor, and D_{Dji} and D_{Dij} are the cross terms. The parameters α_L and α_T (SI unit: m) specify the longitudinal and transverse dispersivities; and u_i (SI unit: m/s) stands for the velocity field components.

In order to facilitate modeling of stratified porous media in 3D, the tensor formulation by Burnett and Frind ([Ref. 10](#)) can be used. Consider a transverse isotropic media, where the strata are piled up in the z direction, the dispersivity tensor components are:

$$\begin{aligned}
D_{Lxx} &= \alpha_1 \frac{u^2}{|\mathbf{u}|} + \alpha_2 \frac{v^2}{|\mathbf{u}|} + \alpha_3 \frac{w^2}{|\mathbf{u}|} \\
D_{Lyy} &= \alpha_1 \frac{v^2}{|\mathbf{u}|} + \alpha_2 \frac{u^2}{|\mathbf{u}|} + \alpha_3 \frac{w^2}{|\mathbf{u}|} \\
D_{Lzz} &= \alpha_1 \frac{w^2}{|\mathbf{u}|} + \alpha_2 \frac{u^2}{|\mathbf{u}|} + \alpha_3 \frac{v^2}{|\mathbf{u}|} \\
D_{Lxy} &= D_{Lyx} = (\alpha_1 - \alpha_2) \frac{uv}{|\mathbf{u}|} \\
D_{Lxz} &= D_{Lzx} = (\alpha_1 - \alpha_3) \frac{uw}{|\mathbf{u}|} \\
D_{Lyz} &= D_{Lzy} = (\alpha_1 - \alpha_3) \frac{vw}{|\mathbf{u}|}
\end{aligned} \tag{5-18}$$

In Equation 5-18 the fluid velocities u , v , and w correspond to the components of the velocity field \mathbf{u} in the x , y , and z directions, respectively, and α_1 (SI unit: m) is the longitudinal dispersivity. If z is the vertical axis, α_2 and α_3 are the dispersivities in the transverse horizontal and transverse vertical directions, respectively (SI unit: m). Setting $\alpha_2 = \alpha_3$ gives the expressions for isotropic media shown in Bear (Ref. 9 and Ref. 11).

Adsorption

As species travel through a porous medium they typically attach to (adsorb), and detach (desorb) from the solid phase, which slows chemical transport through the porous medium. Adsorption and desorption respectively reduces or increases species concentrations in the fluid. The adsorption properties vary between chemicals, so a plume containing multiple species can separate into components (Ref. 6). The Adsorption feature includes four predefined and one user defined relationships to predict the solid concentrations, c_{pi} from the concentration in the liquid phase, c_i :

$$\begin{aligned}
c_P &= c_{P\max} \frac{K_L c}{1 + K_L c} && \text{Langmuir} \\
c_P &= K_F \left(\frac{c}{c_{\text{ref}}} \right)^{N_F} && \text{Freundlich} \\
c_P &= c_{P\max} \frac{b_T c}{(1 + (b_T c)^{N_T})^{1/N_T}} && \text{Toth} \\
c_P &= \frac{K_B c_0 c}{(c_S - c) \left(1 + (K_B - 1) \frac{c}{c_S} \right)} && \text{BET}
\end{aligned} \tag{5-19}$$

The above equations contains the following parameters:

- Freundlich: Freundlich constant K_F (SI unit: mol/kg), Freundlich exponent N_F (dimensionless), and reference concentration c_{ref} (SI unit: mol/m³).
- Langmuir: Langmuir constant K_L (SI unit: m³/mol), and adsorption maximum $c_{P\max}$ (SI unit: mol/kg).
- Toth: Toth constant b_T (SI unit: m³/mol), Toth exponent N_T (dimensionless), and adsorption maximum $c_{P\max}$ (SI unit: mol/kg).
- BET (Braunauer-Emmett-Teller): BET constant K_B (dimensionless), and a monolayer adsorption capacity c_0 (SI unit: mol/kg) and a Saturation concentration, c_S (SI unit: mol/m³).

These predefined expressions are adsorption isotherms that describe the amount of species sorbed to the solid. Defined at equilibrium, the switch between liquid and solid phases is instantaneous.

Using a [Species Source](#) feature, arbitrary expressions can be entered to define, for example, nonequilibrium and temperature-dependent adsorption laws, including those set out by Fetter ([Ref. 7](#)) and Bear and Verruijt ([Ref. 8](#)).

The retardation factor, \mathbf{RF} , describes how adsorption slows the solute velocity, \mathbf{u}_c , relative to the average linear velocity of the fluid, \mathbf{u}_a , as in

$$\mathbf{RF} = 1 + \frac{\rho_b}{\theta} \frac{\partial c_P}{\partial c} = \frac{\mathbf{u}_a}{\mathbf{u}_c}$$

If the contaminant moves at the average linear velocity of the fluid for $\mathbf{RF} = 1$. For $\mathbf{RF} > 1$, the contaminant velocity is smaller than the fluid velocity owing to residence time on solids.

Reactions

Chemical reactions of all types influence species transport in porous media. Examples include biodegradation, radioactive decay, transformation to tracked products, temperature- and pressure-dependent functions, exothermic reactions, and endothermic reactions. The reactions represent change in species concentration per unit volume porous medium per time. Reaction terms are used on the right-hand side of the governing equation to represent these processes. For reactions in a fluid phase, multiply the expression by the fluid volume fraction θ . Similarly, solid phase reaction expressions include the bulk density, ρ_b , and gas phase reactions include the gas volume fraction, α_v .

The following expressions define some common types of reactions:

$$\begin{aligned}
 R_{Li} &= -\theta \frac{\ln 2}{\lambda_{Li}} c_i && \text{Radioactive decay — liquid} \\
 R_{Pi} &= -\rho_b \frac{\ln 2}{\lambda_{Pi}} \left(\frac{\partial c_{Pi}}{\partial c_i} \right) c_i && \text{Radioactive decay — solid} \\
 R_{Gi} &= -\frac{\ln 2}{\lambda_{Gi}} \alpha_v \left(\frac{\partial c_{Gi}}{\partial c} \right) c_i && \text{Radioactive decay — gas} \\
 R_{Lk} &= \theta \zeta_{Li} c_i && \text{Creation from parent } c_{Li} \text{ — liquid} \\
 R_{Pk} &= \rho_b \zeta_{Pi} \left(\frac{\partial c_{Pi}}{\partial c_i} \right) c_i && \text{Creation from sorbed parent } c_{Pi} \text{ — solid} \\
 R_{Gk} &= -\zeta \alpha_v \left(\frac{\partial c_{Gi}}{\partial c} \right) c_i && \text{Reaction — gas}
 \end{aligned}$$

where λ is the chemical half life, ζ is a reaction rate, and the subscripts L , P , and G denote liquid, solid, and gas phases, respectively. In the equations, the reactions either depend on liquid concentration c_i or solid phase concentrations c_{Pi} obtained using the sorption derivative with c_i or gas phase concentration c_{Gi} depending on the gas volume fraction, the volatilization, and the liquid concentration.

Reaction rates can vary with results from other equations in your model, such as temperature. For example, enter the Arrhenius rate law given in [Ref. 13](#):

$$\zeta_T = \zeta_R \exp \left[\frac{E_a (T - T_R)}{R_u T T_R} \right] \quad (5-20)$$

In Equation 5-20, T denotes the current absolute temperature, T_R denotes the reference absolute temperature, E_a is the activation energy, and R_u is the universal gas constant.

Mass Transport in Fractures

When thin fractures occur in porous media, fluid flow tends to move faster along the fracture than in the surrounding media. The transport of chemical species therefore also occur also faster in the direction of the fractures.

The fluid flow in a fracture can be modeled using Darcy's law formulated in a thin sheet of porous medium (a fracture):

$$\mathbf{u} = \frac{\kappa}{\mu} \nabla_t p$$

Here \mathbf{u} is the tangential Darcy velocity, κ is the fracture permeability, μ the fluid's dynamic viscosity, and $\nabla_t p$ is the tangential gradient of the fluid pressure.

The equation to solve for mass transport of species c_i in a thin fracture, embedded in a porous media, is derived from Equation 5-13. The resulting equation is:

$$d_{fr} \left(\frac{\partial \rho_b c_{p,i}}{\partial t} + \frac{\partial \varepsilon_p c_i}{\partial t} + \nabla_t \cdot (D_{e,i} \nabla_t c_i) + \mathbf{u} \cdot \nabla_t c_i \right) = d_{fr} R_i + d_{fr} S_i + n_0 \quad (5-21)$$

Here d_{fr} is the fracture thickness, $c_{p,i}$ the amount of species adsorbed to (or desorbed from) the porous matrix (moles per unit dry weight of the solid), ε_p is the fracture porosity, and D_e is the effective diffusivity. The first two terms on the right hand side represent source terms from reactions, and n_0 corresponds to out-of plane flux from the adjacent porous domain.

In order to arrive at the tangential differential equation, the gradient is split into the contributions normal and tangential to the fracture:

$$\nabla c_i = \nabla_n c_i + \nabla_t c_i$$

The normal gradient is defined in the direction normal to the boundary representing the fracture and the tangential gradient is defined along the boundary. Assuming that the variations in the normal (thin) direction of the fracture are negligible compared to those in the tangential direction, the gradient is simplified as:

$$\nabla c_i = \nabla_t c_i$$

Using [The Transport of Diluted Species in Fractures Interface](#), the transport along fracture boundaries alone is solved for. In this case the transport in the surrounding porous media neglected and the out-of plane flux n_0 vanishes.




See [Fracture](#) for more information about the boundary feature solving Equation 5-21. See [The Transport of Diluted Species in Fractures Interface](#) for more information about the physics interface solving the equation on boundaries only.

References

1. R. Codina, “A discontinuity-capturing crosswind-dissipation for the finite element solution of the convection-diffusion equation”, *Computer Methods in Applied Mechanics and Engineering*, vol. 110, pp. 325–342, 1993.
2. P.V. Danckwerts, “Continuous flow systems: Distribution of residence times”, *Chem. Eng. Sci.*, vol. 2, no. 1, 1953.
3. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 2, 4th ed., Pergamon Press, Oxford, U.K., 1991.
4. J.M. Coulson and J.F. Richardson, *Chemical Engineering*, vol. 1, 4th ed., Pergamon Press, Oxford, U.K., 1991.
5. D.E Rosner, *Transport Processes in Chemically Reacting Flow Systems*, ISBN-13: 978-1483130262, Butterworth-Heinemann, 1986.
6. D.M. Mackay, D.L. Freyberg, P.V. Roberts, and J.A. Cherry, “A Natural Gradient Experiment on Solute Transport in a Sand Aquifer: 1. Approach and Overview of Plume Movement”, *Water Resour. Res.*, vol. 22, no. 13, pp. 2017–2030, 1986.
7. C.W. Fetter, *Contaminant Hydrogeology*, Prentice Hall, 1999.
8. J. Bear and A. Verruijt, *Modeling Groundwater Flow and Pollution*, D. Reidel Publishing, 1994.
9. J. Bear, *Hydraulics of Groundwater*, McGraw-Hill, 1979.
10. R.D. Burnett and E.O. Frind, “An Alternating Direction Galerkin Technique for Simulation of Groundwater Contaminant Transport in Three Dimensions: 2. Dimensionality Effects”, *Water Resour. Res.*, vol. 23, no. 4, pp. 695–705, 1987.

11. J. Bear, *Dynamics of Fluids in Porous Media*, Elsevier Scientific Publishing, 1972.
12. R.J. Millington and J.M. Quirk, “Permeability of Porous Solids”, *Trans. Faraday Soc.*, vol. 57, pp. 1200–1207, 1961.
13. I. Langmuir, “Chemical Reactions at Low Temperatures”, *J. Amer. Chem. Soc.*, vol. 37, 1915.
14. R.B. Bird, W.E. Stewart, and E.N. Lightfoot, *Transport Phenomena*, 2nd ed., John Wiley & Sons, Inc., 2007.

Heat Transfer Interfaces

This chapter has information about the heat transfer interfaces used for modeling, particularly in porous media. As with all other physical descriptions simulated by COMSOL Multiphysics[®], any description of heat transfer can be directly coupled to any other physical process. This is particularly relevant for systems based on chemical reactions and mass transfer along with fluid flow. The following physics interfaces can be found under the **Heat Transfer** branch () when adding a physics interface:

- Heat Transfer in Solids
- Heat Transfer in Fluids
- Heat Transfer in Solids and Fluids
- Conjugate Heat Transfer, Laminar Flow
- Electromagnetic Heating, Joule Heating and Induction Heating
- Thin Structures, Heat Transfer in Fractures
- Heat and Moisture Transport:
 - Building Materials
 - Moist Air
 - Heat and Moisture Flow, Laminar Flow

- Heat Transport in Porous Media
- Local Thermal Nonequilibrium

Some of these interfaces are available with COMSOL Multiphysics and described in the *Comsol Multiphysics Reference Manual*. Others are only available for the Porous Media Flow module and the Heat Transfer module. Please take a look at the Specification chart for a detailed overview of the functionality available in each product (<https://www.comsol.com/products/specifications/>).

In this chapter you will find assistance in selecting the right heat transfer interface for your application by getting a short overview over the different interfaces available. Furthermore, the important mechanisms of heat transfer are introduced. However, a detailed list of available domain and boundary conditions and other features for each interface as well as introductions to the theory of the implemented features are presented in the *Heat Transfer Module User's Guide*. You will find the links to the appropriate chapters in the text below.

Some of the physics interfaces found under the **Heat Transfer** branch are predefined multiphysics interfaces and are therefore described in Chapter [Multiphysics Interfaces and Couplings](#).

In this chapter:

- [Selecting the Right Heat Transfer Interface](#)
- [The Mechanisms for Heat Transfer](#)

Selecting the Right Heat Transfer Interface

The documentation of all features in the Heat Transfer and Heat Transfer in Porous Media interfaces are available in the *Heat Transfer Module User's Guide*. At the end of this section you will find links to the specific sections which are relevant for simulations of heat transfer in porous media.

The Heat Transfer branch included with the Porous Media Flow Module license includes a number of subbranches to describe energy transport.

[The Heat Transfer in Solids Interface](#) provides tools for analyzing heat transfer that is proportional to a temperature gradient, or conduction. The temperature equation defined in solid domains corresponds to the differential form of the Fourier's law. It incorporates boundary and source options to represent transversal fluxes such as convection and radiation at adjacent domains that you do not explicitly model.

[The Heat Transfer in Fluids Interface](#) is used to model heat transfer by conduction, convection, and radiation. The temperature equation defined in fluid domains corresponds to the convection-diffusion equation that may contain additional contributions like heat sources.

A combination of both is provided in the interface [The Heat Transfer in Solids and Fluids Interface](#) and 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. This interface is automatically added if the precoupled multiphysics node Conjugate Heat Transfer is chosen. In this case, the Heat Transfer interface is coupled with a Laminar Flow interface. This is further described in Section [The Conjugate Heat Transfer, Laminar Flow and Turbulent Flow Interfaces](#).

The settings for Heat Transfer in Solids and Fluids are the same as for [The Heat Transfer in Fluids Interface](#).

Heat transfer through conduction and convection (both nonisothermal flow and conjugate heat transfer) in solid and free media is supported by physics interfaces shipped with the basic COMSOL Multiphysics license. However, there are some enhanced features like special boundary conditions which are only available for certain

modules. For a detailed overview of the functionality available in each product, visit <https://www.comsol.com/products/specifications/>.

To get an overview over the functions available with the Porous Media Flow Module the following tables list the available domain and boundary conditions:

In the table below there is a list of domain nodes available in [The Heat Transfer in Fluids Interface](#) if the Porous Media Flow Module is licensed:

- | | |
|---|---|
| • Fluid | • Phase Change Material |
| • Heat Source | • Solid |
| • Initial Values | • Thickness |
| • Moist Air (Heat Transfer Interface) | • Translational Motion |

[The Heat Transfer in Fluids Interface](#) has the following boundary nodes available with the Porous Media Flow module:

- | | |
|--|--|
| • Boundary Heat Source | • Periodic Condition (Heat Transfer Interface) |
| • External Temperature (Thin Layer, Thin Film, Fracture) | • Surface-to-Ambient Radiation (Heat Transfer Interface) |
| • Heat Flux | • Symmetry (Heat Transfer Interface) |
| • Heat Source (Thin Layer, Thin Film, Fracture) | • Temperature |
| • Inflow | • Thermal Insulation |
| • Open Boundary | • Thin Layer (Heat Transfer Interface) and Solid (Heat Transfer in Shells Interface) |
| • Outflow | |

[The Heat Transfer in Fluids Interface](#) has the following edge nodes (3D components only):

- [Line Heat Source](#)

The [Heat Transfer in Fluids Interface](#) has the following point nodes:

- [Point Heat Source](#)

This module also includes [The Heat Transfer in Porous Media Interface](#). The physics interface itself supports heat transfer in porous media where the thermal conductivity and heat capacity are affected by the extent of porosity. This interface therefore offers to choose specific media like a [Porous Medium](#) in general or a [Building Material](#) via the domain nodes.

To model heat transfer by conduction and convection in thin porous structures, like fractures or cracks, [The Heat Transfer in Fractures Interface](#) is available.

The documentation of all features in the Heat Transfer and Heat Transfer in Porous Media interfaces are available in the *Heat Transfer Module User's Guide*, specifically in these sections:



- [The Heat Transfer Module Interfaces](#)
- [The Nonisothermal Flow and Conjugate Heat Transfer Interfaces](#)
- [The Heat Transfer in Solids Interface](#)
- [The Heat Transfer in Fluids Interface](#)
- [The Heat Transfer in Porous Media Interface](#)
- [The Local Thermal Nonequilibrium Interface](#)
- [The Heat Transfer in Fractures Interface](#)
- [Theory for the Heat Transfer Module](#)
- [Theory for the Nonisothermal Flow and Conjugate Heat Transfer Interfaces](#)
- [Theory for Heat Transfer in Porous Media](#)
- [Theory for Heat Transfer in Thin Structures](#)
- [Theory for the Inflow Boundary Condition](#)

A variety of modeling techniques are discussed in the following sections of the *Heat Transfer Module User's Guide*:



-
- [Heat Transfer 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](#)
 - [Solver Settings](#)
 - See also [The Heat Transfer Module Interfaces](#) and [Theory for the Heat Transfer Module](#)
-

The Mechanisms for Heat Transfer

This section includes these topics:

- [Heat Transfer in Porous Media](#)
- [Phase Change](#)

Heat Transfer in Porous Media

Heat transfer in porous media comes in two flavors. As a porous media is made up of a solid matrix filled with a fluid, heat is transported at different rates through the two types of media. If the heat exchange between fluid and matrix occurs practically instantly — on the time scale considered; that is, both temperatures can be assumed to be identical. Thus there is a thermal equilibrium, and 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. [The Heat Transfer in Porous Media Interface](#) provides tools and support for defining an appropriate matrix model.



Performance of a Porous Microchannel Heat Sink: Application Library path
Porous_Media_Flow_Module/Heat_Transfer/porous_microchannel_heat_sink

If the above assumptions cannot be made, meaning that fluid and matrix are not in thermal equilibrium, a two-temperature model is required to simulate heat transfer. [The Local Thermal Nonequilibrium Interface](#) provides tools and support for modeling the heat transfer in the solid matrix and the fluid, and the heat exchange between both phases.



Packed Bed Latent Heat Storage: Application Library path
Porous_Media_Flow_Module/Heat_Transfer/packed_bed_latent_heat_storage

Phase Change

When modeling heat transfer in solids, fluids, and porous media, the properties of a phase-changing material are specified according to the apparent heat capacity

formulation. This formulation gets its name from the fact that the latent heat is included as an additional term in the heat capacity.

The subnode [Phase Change Material](#) is available under the **Solid**, **Fluid**, or **Porous Medium** node.

With phase transitions in porous media, there is often a remaining phase that does not participate in the phase transition. For example, water in the soil, which does not freeze even at values far lower than 0 degrees Celsius, or water in food, which does not evaporate completely during cooking. To consider such residual phases [Immobile Fluids](#) ([Porous Medium](#)) can be used.



Frozen Inclusion: Application Library path

Porous_Media_Flow_Module/Heat_Transfer/frozen_inclusion




Multiphysics Interfaces and Couplings

The Porous Media Flow Module contains several predefined multiphysics interfaces to couple flow in porous media with heat transfer, species and moisture transport, solid mechanics as well as phase transport. When a predefined multiphysics interface is added from the **Model Wizard** or **Add Physics** window, it adds the constituent interfaces and the **Multiphysics** node which automatically includes one or more multiphysics couplings.

In this chapter:

- [Theory for the Poroelasticity Interfaces](#)
- [The Poroelasticity Interfaces](#)
- [Theory for the Poroelasticity, Layered Shell Interface](#)
- [The Poroelasticity, Layered Shell Interface](#)
- [Theory for the Multiphase Flow in Porous Media Interface](#)
- [The Multiphase Flow in Porous Media Interface](#)
- [The Moisture Flow Interface](#)
- [The Heat and Moisture Transport Interfaces](#)
- [The Heat and Moisture Flow Interface](#)

- [The Reacting Flow in Porous Media Interface](#)
- [The Nonisothermal Flow, Brinkman Equations Interface](#)
- [The Two-Phase Flow, Level Set, Brinkman Equations Interface](#)

	The Multiphysics Branch in the <i>COMSOL Multiphysics Reference Manual</i> .
	Coupling features are available from the context menu (right-click the Multiphysics node) or from the Physics toolbar, Multiphysics Couplings menu.
	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.

Theory for the Poroelasticity Interfaces

Small Strain Poroelasticity

Poroelasticity theory describes the interaction between fluid flow and deformation in elastic porous media. This bidirectional multiphysics coupling relates fluid flow and solid deformation with linear constitutive relations.

Of the two constitutive relations governing poroelastic behavior, one relates the stress, strain, and pore pressure:

$$\boldsymbol{\sigma} = \mathbf{C}:\boldsymbol{\varepsilon} - \alpha_B p_f \mathbf{I} \quad (7-1)$$

Here, $\boldsymbol{\sigma}$ is the Cauchy stress tensor, $\boldsymbol{\varepsilon}$ is the strain tensor, α_B is the *Biot-Willis coefficient*, and p_f is the fluid pore pressure. For linear elasticity, the coefficients in the elasticity matrix \mathbf{C} must be measured under “drained” conditions, by measuring the strain induced by a change in stress (or by measuring the induced stress by changes in strain) under constant pore pressure.

By splitting the above relation into volumetric and a deviatoric parts, it can be seen that the deviatoric part (shear stress) is independent of the pore pressure. For an isotropic linear elastic material, this means that

$$\text{dev}(\boldsymbol{\sigma}) = 2G_d \text{dev}(\boldsymbol{\varepsilon})$$

where G_d is the shear modulus of the drained porous matrix.

The volumetric coupling is derived by taking the trace of [Equation 7-1](#)

$$-p_m = K_d \varepsilon_{\text{vol}} - \alpha_B p_f$$

where the mean pressure (positive in compression) is calculated from the trace of the stress tensor $\boldsymbol{\sigma}$ as $p_m = -\text{trace}(\boldsymbol{\sigma})/3$. K_d is the bulk modulus of the drained porous matrix, and the volumetric strain ε_{vol} is a measurement of the porous matrix dilation or contraction (negative in compression).

Note that the **Poroelasticity** multiphysics coupling adds the contribution from the pore pressure, p_f , as a load. In this setting, the stress tensor σ and mean pressure p_m are *effective stress* measures.



The mean pressure is available from the Solid Mechanics interface in the variable `solid.pm` and the volumetric strain in the variable `solid.ev01`. The fluid pore pressure is available from the Darcy's Law interface in the dependent variable `p`.

The other constitutive relation in Biot's theory of poroelasticity ([Ref. 1](#)) relates the increment in fluid content ζ to changes in volumetric strain and pore pressure. The pore pressure is proportional to the dilation of the porous matrix and the variation of fluid content:

$$p_f = M(\zeta - \alpha_B \varepsilon_{vol}) \quad (7-2)$$

Biot and Willis ([Ref. 1](#)) measured the coefficients α_B and M with the unjacketed compressibility test and derived expressions for these coefficients in terms of solid and fluid bulk moduli (or compressibilities).

The variable M , sometimes called the Biot modulus, is the inverse of the storage coefficient S_p in Darcy's law. It is defined through [Equation 7-2](#) as the change in fluid content due to changes in pore pressure under constant deformation (constant volumetric strain):

$$S_p = \frac{1}{M} = \left. \frac{\partial \zeta}{\partial p_f} \right|_{\varepsilon_{ii}}$$

Using this definition, it is possible to measure the storage coefficient S_p directly in the lab, and, in the case of an ideal isotropic porous material, it can be calculated from basic material properties as

$$S_p = \frac{\varepsilon_p}{K_f} + \frac{\alpha_B - \varepsilon_p}{K_s} \quad (7-3)$$

where ε_p is the porosity, K_f the fluid bulk modulus (the inverse of the fluid compressibility χ_f), and K_s the solid bulk modulus; that is, the theoretical bulk modulus of a homogeneous block of the solid material making up the porous matrix.

The Biot–Willis coefficient, α_B , relates the volume of fluid expelled (or sucked into) a porous material element due to the volumetric change of the same element. This

coefficient can be measured experimentally as the change of mean pressure due to changes in the pore pressure under constant deformation (constant volumetric strain). In the case of an ideal isotropic porous material it can be defined in terms of the drained and solid bulk moduli as

$$\alpha_B = \left. \frac{\partial p_m}{\partial p_f} \right|_{\epsilon_{ii}} = 1 - \frac{K_d}{K_s}$$

The drained bulk modulus K_d is always smaller than the solid bulk modulus K_s (a solid block is stiffer than a porous block made of the same material), and therefore the Biot-Willis coefficient is bounded to $\epsilon_p \leq \alpha_B \leq 1$.

The parameter α_B does not depend on the properties of the fluid, but on the properties of the porous matrix. A soft porous matrix is represented by a Biot-Willis coefficient close to unit (since $K_d < K_s$), while for a stiff matrix, α_B is close to the porosity ϵ_p , since $K_d \approx (1 - \epsilon_p)K_s$.

By replacing $K_s = (1 - \alpha_B)/K_d$, the storage coefficient S_p in Equation 7-3 is calculated in terms of the porosity ϵ_p , the Biot-Willis coefficient α_B , and the bulk moduli of the fluid K_f and the drained porous matrix K_d as

$$S_p = \frac{\epsilon_p}{K_f} + (\alpha_B - \epsilon_p) \frac{(1 - \alpha_B)}{K_d} \quad (7-4)$$

For a soft porous matrix, $\alpha_B \approx 1$, or for a stiff porous matrix, $\alpha_B \approx \epsilon_p$, the lower bound for the poroelastic storage is $S_p \approx \epsilon_p/K_f$.

The linear momentum balance for a fully-saturated porous solid in equilibrium under gravitational load reads

$$-\nabla \cdot \sigma = -\nabla(\alpha_B p_f) + \rho_{av} \mathbf{g} \quad (7-5)$$

where σ is the stress tensor computed from linear elasticity, and \mathbf{g} is the gravitational acceleration,. The average density ρ_{av} is computed from

$$\rho_{av} = \rho_f \epsilon_p + \rho_d$$

where ρ_f represents the fluid density and ρ_d the density of the dry porous matrix.

The fluid-to-structure coupling enters as an additional volumetric force in the momentum equation as described in Equation 7-5, which can be identified as an external load due to the fluid pressure, see [External Stress](#) in the *Structural Mechanics Module User's Guide* for details.

Equation 7-5, which describes an equilibrium state (inertial effects are neglected), also applies to the case of a time-dependent flow model. This is a valid assumption in geotechnical applications since the time scale of the structural response is generally many orders of magnitude faster than the time scale of the flow. When you study the coupled process on the time scale of the flow, you can therefore assume that the solid reaches a new equilibrium immediately in response to a change in the flow conditions. This means that the stresses and strains change in time – even if Equation 7-5 appears to be stationary – and that the structure-to-fluid coupling term, involving the rate of strain, is nonzero.

When **Include inertial terms** is selected, the average density is used in the inertial term for the momentum equation. In the presence of a volume force \mathbf{f} (e.g. gravity), the momentum balance then reads

$$\rho_{av} \frac{\partial^2 \mathbf{u}}{\partial t^2} - \nabla \cdot \boldsymbol{\sigma} = -\nabla(\alpha_B p_f) + \mathbf{f} \quad (7-6)$$

where \mathbf{u} represents the solid displacement and ρ_{av} the average density.

Inertial effects are also included when modeling waves in saturated porous media, see the [Theory for the Poroelastic Waves Interfaces](#) section in the *Acoustics Module User's Guide* for details.

Darcy's Law

Darcy's law describes the flow field in the poroelastic medium. The fluid equation comes from the mass conservation

$$\frac{\partial}{\partial t}(\rho_f \epsilon_p) + \nabla \cdot (\rho_f \mathbf{v}) = Q_m \quad (7-7)$$

where ρ_f is the fluid density, ϵ_p is the porosity, and Q_m is a mass sink or source. The fluid velocity is computed from Darcy's law

$$\mathbf{v} = -\frac{\kappa}{\mu} \nabla p_f.$$

where κ is the permeability of the porous matrix, μ is the fluid viscosity, and p_f is the fluid pressure in the pores. If gravity is included, Darcy's velocity is augmented by

$$\mathbf{v} = -\frac{\kappa}{\mu} (\nabla p_f + \rho_f \mathbf{g})$$

where \mathbf{g} is the gravitational acceleration. Note that the fluid velocity is actually the relative velocity of the fluid with respect to the pore walls.

The storage model S_p for linear poroelasticity is derived from

$$\frac{\partial}{\partial t}(\rho_f \epsilon_p) = \rho_f S_p \frac{\partial p_f}{\partial t}$$

where the poroelastic storage term S_p is calculated from [Equation 7-4](#).

For small strain poroelasticity, the mass source or sink Q_m reads

$$Q_m = -\rho_f \alpha_B \frac{\partial}{\partial t} \epsilon_{vol}$$

where $\partial \epsilon_{vol} / \partial t$ is the rate of change in volumetric strain (of the porous matrix), ρ_f is the fluid density, and α_B is the Biot–Willis coefficient. You can interpret the right-hand-side term as the rate of expansion of the pore space. As ϵ_{vol} increases with time, the volume fraction available for the fluid increases and thereby gives rise to a fluid sink; as indicated by the negative sign in the source term Q_m .

The poroelastic storage, together with the expression for the mass source, and the definition of Darcy’s velocity translate the mass conservation equation, [Equation 7-7](#), into

$$\rho_f S_p \frac{\partial p_f}{\partial t} + \nabla \cdot (\rho_f \mathbf{v}) = -\rho_f \alpha_B \frac{\partial}{\partial t} \epsilon_{vol} \quad (7-8)$$

Large Strain Poroelasticity

When a porous body experiences large deformations (large strains or large rotations), the infinitesimal theory cannot be used. In a large deformation context, the momentum equation, [Equation 7-6](#), reads

$$\rho_{av} \frac{\partial^2 \mathbf{u}}{\partial t^2} - \nabla \cdot \mathbf{P}^T = \mathbf{f} \quad (7-9)$$

where the gradient $\nabla = \partial / \partial \mathbf{X}$ corresponds to the material gradient with respect to the undeformed configuration, and \mathbf{P} is the nominal (first Piola–Kirchoff) stress tensor. See [Equation of Motion](#) in the *Structural Mechanics Theory* chapter for details.

The pore pressure contribution enters as a load in [Equation 7-9](#). The first Piola–Kirchoff load generated from the pore pressure reads

$$P_p = \alpha_B p_f J F^{-T} \quad (7-10)$$

Darcy's law is also written with respect to the undeformed configuration. Darcy's velocity in the undeformed configuration reads

$$\mathbf{V} = -\frac{\kappa}{\mu} \nabla p_f$$

where, κ is the permeability of the undeformed porous matrix, μ is the fluid viscosity, and p_f is the fluid pressure in the pores. The symbol $\nabla = \partial/\partial \mathbf{X}$ corresponds to the gradient with respect to the undeformed configuration.

Darcy's velocity in the current (deformed) configuration is then computed with the so-called *Piola transformation* (Ref. 2)

$$\mathbf{v} = \frac{1}{J} \mathbf{F} \cdot \mathbf{V}$$

where \mathbf{F} is the deformation gradient, and J the volume change.

The mass conservation, Equation 7-8, is then written as (Ref. 2)

$$\rho_f S_p \frac{\partial p_f}{\partial t} + \frac{1}{J} \nabla \cdot (\rho_f \mathbf{V}) = -\rho_f \alpha_B \frac{\partial}{\partial t} \epsilon_{vol} \quad (7-11)$$

where the rate of volumetric strain is now computed from Hencky's strain

$$\frac{\partial}{\partial t} \epsilon_{vol} = \frac{\partial}{\partial t} \log J = \frac{1}{J} \frac{\partial J}{\partial t}$$

For large strain poroelasticity, the poroelastic storage is computed from the current porosity, and not from the porosity of the undeformed porous matrix. The porosity of the deformed porous matrix, ϕ , is related to the porosity of the undeformed material ϵ_p as

$$\phi = \frac{1}{J} \epsilon_p$$

so the poroelastic storage (Equation 7-3) in a large strain context reads

$$S_p = \frac{\phi}{K_f} + \frac{\alpha_B - \phi}{K_s} \quad (7-12)$$

For numerical reasons, and also since the solid mechanics equations are solved in a *total Lagrangian formulation* (see [Lagrangian Formulation](#) for details), it is more

efficient to write the mass conservation equation for the fluid in the undeformed configuration. This is done by multiplying [Equation 7-11](#) by the volume change J , so it reads

$$\rho_f S_p J \frac{\partial p_f}{\partial t} + \nabla \cdot (\rho_f \mathbf{V}) = -\rho_f \alpha_B \frac{\partial J}{\partial t} \quad (7-13)$$


In order to track large deformations in the porous matrix, the **Poroelasticity, Large Deformations** multiphysics interface also adds an **Elastic Predeformation** node to the Solid Mechanics interface. See [Elastic Predeformation](#) for details.


Reference for the Poroelasticity Interface

1. M.A. Biot, “Mechanics of Deformation and Acoustic Propagation in Porous Media,” *J. Appl. Phys.*, vol. 33, pp. 1482–1498, 1962.
2. O. Coussy. Poromechanics. John Wiley & Sons, 2004.

The Poroelasticity Interfaces

The Poroelasticity, Solid Interface

The **Poroelasticity, Solid** interface () combines a transient formulation of Darcy’s law with a quasistatic formulation of Solid Mechanics. The coupling occurs on the domain level, where the pore pressure from the Darcy’s Law interface acts as a load for the Solid Mechanics interface, causing swelling or shrinking. Changes in volumetric strain affect the pore space, acting as a mass source or sink in Darcy’s Law.

When a predefined **Poroelasticity, Solid** interface is added from the **Structural Mechanics** branch () of the **Model Wizard** or **Add Physics** windows, **Solid Mechanics** and **Darcy’s Law** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the **Poroelasticity** multiphysics coupling.

On the Constituent Physics Interfaces

- The Solid Mechanics interface is described in the [The Solid Mechanics Interface](#).
- The Darcy’s Law interface is described in [The Darcy’s Law Interface](#).



To model orthotropic or anisotropic porous materials requires the Structural Mechanics Module. For information about the constitutive equations and theory background, see [Structural Mechanics Theory](#).



[Theory for the Poroelasticity Interfaces](#).

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, for example **Poroelasticity**, 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 **Solid Mechanics** and **Darcy’s Law** interfaces are added, an empty **Multiphysics** node appears in the model tree. You can choose from the available coupling features, but the settings in the constituent interfaces are not modified.




	Coupling features are available from the context menu (right-click the Multiphysics node) or from the Physics toolbar, Multiphysics menu.
---	--

TABLE 7-1: MODIFIED SETTINGS FOR A POROELASTICITY INTERFACE

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS
Solid Mechanics	For the Solid Mechanics interface, under Structural Transient Behavior the Structural transient behavior is set to Quasistatic .
Darcy’s Law	For the Darcy’s Law interface, a Poroelastic storage model is added by default.
Poroelasticity	The Domain Selection is the same as that of the participating physics interfaces. The corresponding Solid Mechanics and Darcy’s Law interfaces are preselected in the Coupled Interfaces section.

	<i>Failure of a Multilateral Well</i> : Application Library path Subsurface_Flow_Module/Flow_and_Solid_Deformation/multilateral_well
--	--

Poroelasticity (Multiphysics Coupling)

The **Poroelasticity** multiphysics coupling node () links bidirectionally the Solid Mechanics and Darcy’s Law interfaces to account for poroelastic deformation.

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 poro1.

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 poroelasticity coupling, or select **All domains** as needed.

When **Poroelasticity** is added as an effect of adding a Poroelasticity interface, the selection is the same as for the participating physics interfaces.

Only domains that are active in the physics interfaces selected in the [Coupled Interfaces](#) section can be selected.

POROELASTIC COUPLING PROPERTIES

In the **Reference pressure level** field, enter a reference pressure p_{ref} (SI unit: Pa) at which the pore pressure does not affect the porous matrix deformation. The default value is 1 atm.

For the **Biot–Willis coefficient** list, the default is to take its value **From material**. Choose **User defined** to enter a Biot–Willis coefficient α_B (dimensionless) in the text field.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Solid mechanics** and **Flow in porous media** lists include all applicable physics interfaces.

The default values depend on how the coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (macOS 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 selected 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 Poroelasticity node from a physics interface. If the physics interface is removed from the **Model Builder**, for

example **Darcy's Law** is deleted, then the **Flow in porous media** 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 **Solid mechanics** or **Flow in porous media** 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*.

The Poroelasticity, Large Deformation Interface

The multiphysics interface **Poroelasticity, Large Deformation** couples the Darcy's Law interface and the Solid Mechanics interface. Although the theory for the two Poroelastic interfaces **Poroelasticity, Solid** and **Poroelasticity, Large Deformation** quite differs as shown in [Theory for the Poroelasticity Interfaces](#), the interface settings and the multiphysics interfaces look quite similar. The only obvious difference is that for large deformations, an **Elastic Predeformation** node within the Solid Mechanics interface is active on all domains to track large deformations in the porous matrix. This feature forces the check box **Include geometric nonlinearity** in the solver settings to be checked.

For more information about the interface settings and the settings for the multiphysics node, see [The Poroelasticity, Solid Interface](#) and [Poroelasticity \(Multiphysics Coupling\)](#).



- [Theory for the Poroelasticity Interfaces](#)
- [Elastic Predeformation](#) in the *Structural Mechanics Module User's Guide*
- [Modeling Geometric Nonlinearity](#) in the *Structural Mechanics Module User's Guide*



The Multiphysics interface **Poroelasticity, Large Deformation** 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/>

Theory for the Poroelasticity, Layered Shell Interface


The Theory for the Poroelasticity, Layered Shell interface is basically the same as described in [Theory for the Poroelasticity Interfaces](#). The main difference is the 2D approximation for layered materials. Theoretical background about the 2D approximation is given in the theory chapters of the two coupled interfaces Layered Shell and Layered Darcy's Law.



All relevant information can be found in the following chapters:

- [Theory for the Poroelasticity Interfaces](#).
- [Theory for the Layered Darcy's Law Interface](#).
- [Theory for the Layered Shell Interface](#) in the *Structural Mechanics Module User's Guide*.

The Poroelasticity, Layered Shell Interface

The **Poroelasticity, Layered Shell** interface () combines the **Layered Darcy's Law** formulation with the **Layered Shell** interface for structural mechanics. This multiphysics interface is only available for 3D simulations and is only active on all boundaries where a layered material is defined.




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.

This multiphysics interface is only available when the Structural Mechanics Module, the Composite Materials Module and the Porous Media Flow Module are licensed. For a detailed overview of the functionality available in each product visit <https://www.comsol.com/products/specifications/>.

The coupling occurs on the boundary level, where the pore pressure from the Layered Darcy's Law interface acts as a load for the Layered Shell interface, causing swelling or shrinking. Changes in volumetric strain affect the pore space, acting as a mass source or sink in Darcy's Law.

When a predefined **Poroelasticity, Layered Shell** interface is added from the **Structural Mechanics** branch () of the **Model Wizard** or **Add Physics** windows, **Layered Shell** and **Layered Darcy's Law** interfaces are added to the Model Builder.

In addition, the **Multiphysics** node is added, which automatically includes the **Layered Poroelasticity** multiphysics coupling.



For the settings of each of the coupled interfaces see

- [The Layered Darcy’s Law Interface](#) and
- [The Layered Shell Interface](#) in the *Structural Mechanics Module User’s Guide*.

SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, as in this case **Poroelasticity**, **Layered Shell**, 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 **Layered Shell** and **Layered Darcy’s Law** interfaces are added, an empty **Multiphysics** node appears in the model tree. You can choose from the available coupling features, but the settings in the constituent interfaces are not modified.




Coupling features are available from the context menu (right-click the **Multiphysics** node) or from the **Physics** toolbar, **Multiphysics** menu.

TABLE 7-2: MODIFIED SETTINGS FOR A POROELASTICITY INTERFACE

PHYSICS INTERFACE OR COUPLING FEATURE	MODIFIED SETTINGS
Layered Shell	For the Layered Shell interface, under Structural Transient Behavior the Structural transient behavior is set to Quasistatic .
Layered Darcy’s Law	For the Layered Darcy’s Law interface, under Discretization, the shape function order for the temperature is set to Linear . Also, a Poroelastic Storage model is added by default.
Layered Poroelasticity	The corresponding Layered Shell and Layered Darcy’s Law interfaces are preselected in the Coupled Interfaces section.

Layered Poroelasticity (Multiphysics Coupling)

The **Layered Poroelasticity** multiphysics coupling node () links bidirectionally the Layered Shell and Layered Darcy's Law interfaces to account for poroelastic deformation on layered shells.

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** is **poro**, followed by the number of defined multiphysics couplings, that is **poro1** for the first multiphysics coupling feature in the model.

BOUNDARY SELECTION

When **Layered Poroelasticity** is added to the **Multiphysics** node as an effect of adding a Poroelasticity, Layered Shell interface, the selection is the same as for the participating physics interfaces. In the Boundary Selection menu you can choose between **Manual** and **All boundaries**, the latter being the default option. Note that only boundaries that are active in the physics interfaces selected in the **Coupled Interfaces** section can be selected.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Layered shell** and **Flow in porous media** lists include all applicable physics interfaces.

The default values depend on how the coupling node is created.

- If it is added from the **Physics** ribbon (Windows users), **Physics** contextual toolbar (macOS 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 selected 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 Poroelasticity node from a physics interface. If the physics interface is removed from the **Model Builder**, for

example **Darcy's Law** is deleted, then the **Flow in porous media** 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 **Layered shell** or **Flow in porous media** 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*.

SHELL PROPERTIES

Here you select on which layers of a layered material the physics interface should be active. By default, the **Use all layers** check box is selected. This means that all layers in all layered materials on the selected boundaries are used.

If you clear the **Use all layers** check box, you can select individual layers within a single layered material. This means that the physics interface is restricted to the boundaries on which a specific layered material is defined.



For a general description of layer selections, see [The Shell Properties and Interface Selection Sections](#) in the *Structural Mechanics Module User's Guide*.

POROELASTIC COUPLING PROPERTIES

In the **Reference pressure level** field, enter a reference pressure p_{ref} (SI unit: Pa) at which the pore pressure does not affect the porous matrix deformation. The default value is 1 atm.

For the **Biot–Willis coefficient** list, the default is to take its value **From material**. Choose **User defined** to enter a Biot–Willis coefficient α_B (dimensionless) in the text field.

Theory for the Multiphase Flow in Porous Media Interface

The model equations that are solved in the **Multiphase Flow in Porous Media** interface are based on the mass conservation of each phase and on an extended Darcy's law. The mass conservation equation for each phase is given by

$$\frac{\partial}{\partial t}(\epsilon_p \rho_i s_i) + \nabla \cdot (\rho_i \mathbf{u}_i) = Q_i \quad (7-14)$$

where ϵ_p (dimensionless) is the porosity, and the vector \mathbf{u}_i should be interpreted as the volumetric flux of phase i (SI unit $\text{m}^3/(\text{m}^2 \cdot \text{s})$ or m/s). The volumetric fluxes are determined using the extended Darcy's law ([Ref. 2](#))

$$\mathbf{u}_i = -\frac{\kappa_{ri}}{\mu_i} \kappa (\nabla p_i - \rho_i \mathbf{g}) \quad (7-15)$$

where κ denotes the permeability (SI unit: m^2) of the porous medium, \mathbf{g} the gravitational acceleration vector (SI unit m/s^2), and μ_i the dynamic viscosity (SI unit: $\text{kg}/(\text{m} \cdot \text{s})$), p_i the pressure field (SI unit: Pa), and κ_{ri} the relative permeability (dimensionless) of phase i , respectively. The phase pressures p_i are related through the capillary pressure functions $p_{c_{ij}}$:

$$p_i - p_j = p_{c_{ij}}(s_1, \dots, s_N) \quad i \neq j \quad (7-16)$$

One phase pressure can be chosen independently so that $N - 1$ capillary pressure relations are needed to define the other phase pressures. In addition it is assumed that all phases together fill the pore space completely, so that we have

$$\sum_{i=1}^N s_i = 1 \quad (7-17)$$

Pressure-Saturation Formulation

The algebraic relations in [Equation 7-16](#) and [Equation 7-17](#) allow for a reduction of the number of dependent variables. An often used way is to eliminate $N - 1$ phase pressures and one of the saturations. This results in a so-called pressure-saturation

formulation (Ref. 3). The procedure in the **Multiphase Flow in Porous Media** interface is to pick one of the phases, let us say phase i_c , then express the volume fraction, s_{i_c} , of this phase in terms of the volume fractions of the other phases and in addition to use the pressure, p_{i_c} , of this phase to define the other phase pressures:

$$s_{i_c} = 1 - \left(\sum_{i=1, i \neq i_c}^N s_i \right) \quad (7-18)$$

$$p_i = p_{i_c} + p_{c_{ij}}(s_1, \dots, s_N) \quad \text{for } i \neq i_c \quad (7-19)$$

The equations for the volume fractions s_i , ($i \neq i_c$) are given by Equation 7-14 and Equation 7-15. To arrive at an equation for p_i , the conservation equations of all phases are summed. This results in

$$\frac{\partial}{\partial t} \left(\epsilon_p \sum_{i=1}^N \rho_i s_i \right) + \nabla \cdot \left(\sum_{i=1}^N \rho_i \mathbf{u}_i \right) = Q_{\text{tot}} \quad (7-20)$$

where the total mass source Q_{tot} is given by

$$Q_{\text{tot}} = \sum_{i=1}^N Q_i \quad (7-21)$$

The equations for the volume fractions s_i are solved in the **Phase Transport in Porous Media** interface (see Equation 6-99 and Equation 6-100). The Equation 7-20 for the pressure field p_{i_c} , needed as an input to the **Phase Transport in Porous Media** interface, is solved for in the **Darcy's Law** interface: the **Multiphase Flow in Porous Media** multiphysics coupling interface replaces the equation

$$\frac{\partial}{\partial t} (\epsilon_p \rho_{i_c}) + \nabla \cdot (\rho_{i_c} \mathbf{u}_d) = Q_m \quad \text{where} \quad \mathbf{u}_d = -\frac{1}{\mu_{i_c}} \kappa (\nabla p_{i_c} - \rho_{i_c} \mathbf{g}) \quad (7-22)$$

which is originally implemented in the **Darcy's Law** interface, with Equation 7-20 by adding the following terms to the left-hand side of Equation 7-22:

$$\frac{\partial}{\partial t} \left(\epsilon_p \sum_{i=1}^N \rho_i s_i - \epsilon_p \rho_{\text{tot}} \right) + \nabla \cdot \left(\sum_{i=1}^N \rho_i \mathbf{u}_i - (\rho_{\text{tot}} \mathbf{u}_d) \right) \quad (7-23)$$

and by setting the right-hand side Q_m to be equal to Q_{tot} .

Furthermore, the **Multiphase Flow in Porous Media** multiphysics coupling interface couples the user inputs for the porosity and permeability of the porous matrix in the **Darcy's Law** interface to the corresponding user input fields in the **Phase and Porous Media Transport Properties** feature, and couples the pressure field computed for in the **Darcy's Law** interface to the user input field for the pressure of the phase computed from the volume constraint.

Note that when the hydraulic conductivity of a domain is specified in the **Darcy's Law** interface (instead of the permeability), the coupled **Phase and Porous Media Transport Properties** feature is supplied with a permeability that is computed using a reference kinematic viscosity of $1.004 \cdot 10^{-6} \text{ m}^2 \text{ s}^{-1}$ (kinematic viscosity of water at 293.15 K).

In the other direction, the multiphysics coupling node provides the averaged density (denoted by ρ_{tot} in Equation 7-23 above) and effective viscosity to the **Darcy's Law** interface.

Mass Source

When adding a Mass Source node in the coupled Phase Transport interface, it is by default assumed that the net mass source of all phases, Q_{tot} , equals zero. This implies

$$Q_{i_c} = - \sum_{i=1, i \neq i_c}^N Q_i \quad (7-24)$$

However, when mass is transferred from or to phases not accounted for in the phase transport interface — for instance, when mass is released or absorbed by the solid part of the porous matrix — select the **Mass transfer to other phases** check box. In this case the mass source for the phase calculated from the volume constraint should also be specified, and the net total mass source Q_{tot} is supplied as a right-hand side to the Darcy Equation 7-20 for p_{i_c} .

Boundary Conditions

When supplying boundary conditions for the coupled **Phase Transport in Porous Media** and **Darcy's Law** interfaces, bear in mind that the **Pressure** boundary condition in the **Darcy's Law** interface affects the pressure of the phase from the volume constraint, p_{i_c} . If the pressure of another phase needs to be supplied, then take into account the capillary pressure.


Since the coupled **Darcy's Law** interface uses the conservation equation for the total mass (Equation 7-20) to solve for p_{i_c} , the **Mass Flux** boundary condition in the **Darcy's Law** interface should be used to prescribe the mass flux of all phases together.


If the mass flux of the phase computed from the volume constraint needs to be prescribed at a boundary, and the total mass flux is not known a priori, it might be necessary to compute for the total mass flux in the form of a Lagrange multiplier by using the weak constraint formulation for the pressure boundary condition in the **Darcy's Law** interface.

References

1. J. Bear, *Dynamics of Fluids in Porous Media*, Elsevier Scientific Publishing, 1972.
2. Z. Chen, G. Huan, and Y. Ma, *Computational Methods for Multiphase Flows in Porous Media*, Philadelphia: Society for Industrial and Applied Mathematics, 2006.
3. R. Helmig, *Multiphase Flow and Transport Processes in the Subsurface – A Contribution to the Modeling of Hydrosystems*, Springer-Verlag, 1997.

The Multiphase Flow in Porous Media Interface

The **Multiphase Flow in Porous Media** interface () combines the **Darcy's Law** interface with the **Phase Transport in Porous Media** interface to model the flow and transport of multiple immiscible phases in a porous medium.

When a predefined **Multiphase Flow in Porous Media** interface is added from **Porous Media and Subsurface Flow** branch () of the **Model Wizard** or **Add Physics** window, **Phase Transport** and **Darcy's Law** interfaces are added to the Model Builder. In addition, the **Multiphysics** node is added, which automatically includes the **Multiphase Flow in Porous Media** multiphysics coupling.

On the Constituent Physics Interfaces

The **Phase Transport in Porous Media** interface is used to simulate the transport of immiscible multiple species in a porous medium and solves for their averaged volume fractions (also called saturations). The interfaces between the different phases are not tracked explicitly, although microscopic interfacial effects are taken into account in the macroscopic equations through the capillary pressure functions.

The **Darcy's Law** interface is used to simulate fluid flow through interstices in a porous medium. It can be used to model low-velocity flows or media where the permeability and porosity are very small, and for which the pressure gradient is the major driving force and the flow is mostly influenced by the frictional resistance within the pores.


SETTINGS FOR PHYSICS INTERFACES AND COUPLING FEATURES

When physics interfaces are added using the predefined couplings, for example **Multiphase Flow in Porous Media**, 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 **Phase Transport** and **Darcy's Law** interfaces are added, an empty **Multiphysics** node appears in the model tree. You can choose from the available coupling features but the settings in the constituent interfaces are not modified.

Multiphase Flow in Porous Media (Multiphysics Coupling)

The **Multiphase Flow in Porous Media** multiphysics coupling node () links bidirectionally the **Phase Transport in Porous Media** and **Darcy's Law** interfaces.

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 `mfpm1`.

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 multiphase flow coupling, or select **All domains** as needed.

When **Multiphase Flow in Porous Media** is added as an effect of adding a Multiphase Flow in Porous Media interface, the selection is the same as for the participating physics interfaces.

Only domains that are active in the participating physics interfaces (which are chosen from the **Coupled Interfaces** selection list) can be selected.

COUPLED INTERFACES

This section defines the physics involved in the multiphysics coupling. The **Phase transport** and **Flow in porous media** lists include all applicable physics interfaces.

The default values depend on how the 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 selected 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 Multiphase Flow in Porous Media node from a physics interface. If the physics interface is removed from the **Model Builder**, for example **Darcy's Law** is deleted, then the **Flow in porous media** list defaults to **None** as there is nothing to couple to.




Please find related information under

- [The Phase Transport Interfaces](#) and
- [Theory for the Phase Transport Interfaces](#)

in the *CFD Module User's Guide*.

Well (Multiphysics Coupling)

The **Multiphase Flow in Porous Media** () interface contains an optional **Well** multiphysics coupling feature on points (in 2D) or edges (in 3D). The **Well** coupling node is intended to model injection or production wells and links bidirectionally the **Phase Transport in Porous Media** and **Darcy's Law** interfaces.

The Subsurface Flow Module license is required to use this coupling feature (see <https://www.comsol.com/products/specifications/>).

SETTINGS

The **Label** is the default multiphysics coupling feature name.

The default **Name** (for the first **Well** multiphysics coupling feature in the model) is wellmpp1.

POINT OR EDGE SELECTION

When nodes are added from the context menu, you can select **Manual** (the default) from the **Selection** list to choose specific points or edges, or select **All points** or **All edges** as needed.

COUPLED INTERFACES

This section controls which individual interfaces are coupled by the current coupling feature. 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 correct physics interface again from the **Phase transport** or **Flow in porous media** lists

WELL

Select whether the well is a production or injection well: for an injection well positive mass fluxes signify inflowing fluxes and for a production well positive mass fluxes are outflowing fluxes.

Either specify the total **Total mass flow rate** M_0 (SI unit: kg/s), the **Total mass flow rate per unit length** M_1 (SI unit: kg/(m·s)), the **Pressure** p_0 (SI unit: Pa), the **Pressure head** H_{p0} (SI unit: m), or the **Hydraulic head** H_0 (SI unit: m). The elevation D used to convert the hydraulic head to a pressure, is defined by the gravity effects in the coupled Darcy's Law interface.

Select the **Specify well diameter** check box to define the **Well diameter** d_w (SI unit: m). This averages the mass source on the boundary of a cylinder around the line. This option avoids obtaining an increasing pressure value at the line when meshing finer than this diameter. It makes use of the `circavg` operator.

If the well diameter is defined, the option to specify the **Total mass flux** N_0 (SI unit: kg/(m²·s)) is also available.

PHASES

The number of **Phase** sections depends on the number of phases defined in the coupled Phase Transport in Porous Media interface: the number of sections is equal to the number of phases not computed from the volume constraint, which is in turn equal to the number of phases defined in the coupled Phase Transport in Porous Media interface minus one.

Specify the boundary condition for each phase not computed from the volume constraint. You can choose to enter a value for the **Mass flow rate** $M_{0,i}$ (SI unit: kg/s), the **Mass flow rate per unit length** $M_{1,i}$ (SI unit: kg/(m·s)), or the **Volume fraction** $s_{0,i}$ (dimensionless). If the well diameter is specified, the option to specify the **Mass flux** $N_{0,i}$ (SI unit: kg/(m²·s)) is also available. For a production well, you can additionally choose the **Outflow** option. For an injection well you can additionally choose to specify the **Fraction of total volumetric flow rate** $s_{0,i}$ (dimensionless).

Fraction of total volumetric flow rate

When the **Fraction of total volumetric flow rate** $s_{0,i}$ is specified for a phase, a mass flow rate per unit length $M_{1,i}$ is supplied for that phase, which is given by

$$M_{1,i} = \rho_i s_{0,i} V_1 \quad (7-25)$$

where V_1 is the total volumetric flow rate per unit length, which is computed as follows: assume that there are N phases, and that phase 1 is the phase computed from the

volume constraint, that for phases i , for $i = 2, \dots, p$, the fraction of total volumetric flow rate $s_{0,i}$ is supplied, and that for the phases i , for $i = p+1, \dots, N$, the mass flow rate per unit length $M_{1,i}$ is given. Then V_1 can be computed from the relation

$$V_1 = \frac{M_{1,1}}{\rho_1} + \sum_{i=2}^p V_1 s_{i,0} + \sum_{i=p+1}^N \frac{M_{1,i}}{\rho_i} \quad (7-26)$$

where the mass flow rate $M_{1,1}$ for the phase computed from the volume constraint is given by

$$M_{1,1} = M_1 - \sum_{i=2}^p V_1 \rho_i s_{i,0} - \sum_{i=p+1}^N M_{1,i} \quad (7-27)$$

Note that the total mass flow rate per unit length M_1 is readily available when any of the **Total mass flow rate**, **Total mass flow rate per unit length**, or the **Total mass flux** options is chosen. Furthermore, the total mass flow rate per unit length M_1 is also available as a Lagrange multiplier in case any of the **Pressure**, the **Pressure head**, or the **Hydraulic head** is given, as these conditions are implemented as weak constraints. For the mass flow rates per unit length $M_{1,i}$ of the individual phases, it holds similarly that they are readily available when any of the **Mass flow rate**, **Mass flow rate per unit length**, or the **Mass flux** options are chosen, and that for the **Volume fraction** option, implemented as a weak constraint, the mass flow rate per unit length $M_{1,i}$ is computed as a Lagrange multiplier.

Outflow


When the **Outflow** option is specified for a phase s_i , an outflowing mass flow rate per unit length $M_{1,i}$ is supplied for that phase, which is given by

$$M_{1,i} = \frac{\rho_i \kappa_{ri}}{\lambda_r \mu_i} M_1 \quad \text{with} \quad \lambda_r = \sum_{i=1}^N \frac{\rho_i \kappa_{ri}}{\mu_i} \quad (7-28)$$

where κ_{ri} denotes the relative permeability of phase i .

The Moisture Flow Interface

The Moisture Flow interfaces model moisture transport in air by vapor diffusion and convection.

When a Moisture Flow () multiphysics interface is added from the **Chemical Species Transport>Moisture Flow** branch of the **Model Wizard** or **Add Physics** windows, the **Single-Phase Flow** interface Laminar 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.

Although the Moisture Flow interface can be found under the **Chemical Species Transport** branch, it has been implemented in the Heat Transfer Module and made available for the Porous Media Flow Module. Therefore, detailed information is available in the *Heat Transfer Module User's Guide*.



- [The Moisture Flow Interfaces](#)
 - [Theory for the Moisture Flow Interface](#)
-

The Heat and Moisture Transport Interfaces

The Heat and Moisture Transport interfaces couple heat transfer and moisture transport in building materials and moist air.

The Building Materials version of the Heat and Moisture Transport multiphysics interface combines the Heat Transfer in Building Materials and Moisture Transport in Building Materials interfaces. A Heat and Moisture multiphysics node is automatically added to couple these interfaces. It takes in account heat and moisture storage, latent heat effects, and liquid and convective transport of moisture. In addition, the temperature dependency of the moisture properties is automatically accounted for.

When the Moist Air version of the predefined Heat and Moisture Transport interface is added, it combines the Heat Transfer in Moist Air and 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.



The versions of the Heat and Moisture Transport multiphysics interface are described in the *Heat Transfer Module's User's Guide*:

- [The Building Materials Version of the Heat and Moisture Transport Multiphysics Interface](#)
- [The Moist Air Version of the Heat and Moisture Transport Multiphysics Interface](#)

The Heat and Moisture Flow Interface

The Heat and Moisture Flow interface available with the Porous Media Flow Module model heat and moisture transport in air by laminar flow.

When a Heat and Moisture Flow multiphysics interface is added from the **Heat Transfer>Heat and Moisture Transport>Heat and Moisture Flow** branch of the **Model Wizard** or **Add Physics** windows, a Laminar Flow, a Heat Transfer in Moist Air interface, and a Moisture Transport in air interface are added to the Model Builder.


In addition, the **Multiphysics** node is added, which includes the **Moisture Flow**, **Heat and Moisture**, and **Nonisothermal Flow** multiphysics coupling features.



The documentation for this interface is available in the *Heat Transfer Module User's Guide*

- [The Heat and Moisture Flow Interfaces](#)
- [Theory for the Heat and Moisture Flow Interfaces](#)

The Reacting Flow in Porous Media Interface


The Reacting Flow in Porous Media, Transport of Diluted Species interface () is used to study the flow and chemical composition of a gas or liquid moving through the interstices of a porous medium.

It combines the Brinkman Equations and Transport of Diluted Species in Porous Media interfaces. The Reacting Flow, Diluted Species multiphysics coupling feature, which is added automatically, couples the fluid flow and mass transport.

The interface can be used for stationary and time-dependent analysis in 2D, 2D axial symmetry, and 3D.

Detailed information is available in the *Chemical Reaction Engineering User's Guide* in the section [The Reacting Flow in Porous Media Multiphysics Interfaces](#).

The Nonisothermal Flow, Brinkman Equations Interface

The Multiphysics coupling **Nonisothermal Flow, Brinkman Equations** () is used to model heat transfer and flow in porous media where both the temperature and flow fields have to be coupled. Here, the nonlocal couplings between porous media flow and heat transfer are set up automatically.

The Multiphysics coupling interface can be used for stationary and time-dependent analysis in 2D, 2D axial symmetry, and 3D.

More detailed information is available in the *Subsurface Flow Module User's Guide* in the section [The Nonisothermal Flow, Brinkman Equations Interface](#).



-
- [The Nonisothermal Flow, Brinkman Equations Interface](#) in the *Subsurface Flow Module User's Guide*
 - [The Brinkman Equations Interface](#)
 - [The Heat Transfer in Porous Media Interface](#) in the *Heat Transfer Module User's Guide*
 - [The Nonisothermal Flow and Conjugate Heat Transfer, Laminar Flow, Turbulent Flow and LES Interfaces](#) in the *CFD Module User's Guide*
 - [Heat Transfer and Fluid Flow Coupling](#) in the *Heat Transfer Module User's Guide*
 - [The Multiphysics Branch](#) in the *COMSOL Multiphysics Reference Manual*.
-

The Two-Phase Flow, Level Set, Brinkman Equations Interface

This Multiphysics interface was designed to track the interface between two immiscible fluids within a porous medium as for example when simulating resin transfer molding.

The tracking is done using the Level Set equation, and the velocity field for this equation can be calculated from the Brinkman Equations.

The immiscible fluids can be incompressible or compressible. The interface is available in 2D, 2D axial symmetric, and 3D.

When adding the **Two-Phase Flow, Level Set, Brinkman Equations** multiphysics interface, a **Brinkman Equations** and a **Level Set** interface are added to the **Model Builder**. In addition, the multiphysics coupling feature **Two-Phase Flow, Level Set** is created automatically under the **Multiphysics** node (see [The Two-Phase Flow, Level Set Coupling Feature](#) in the *CFD Module User's Guide*). The **Wetted Wall** coupling feature is also added, but with empty selection. Other **Wetted Wall** multiphysics coupling features and the **Interior Wetted Wall** coupling feature can be added when needed to model walls. For more information about these coupling features read [The Wetted Wall Coupling Feature](#) and [The Interior Wetted Wall Coupling Feature](#) in the *CFD Module User's Guide*.

Simulations using the **Two-Phase Flow, Level Set, Brinkman** equations interface are always time-dependent since the position of an interface is almost always dependent of its history.

Note that this multiphysics coupling does not include capillary effects arising inside the porous structure itself as these effects can be neglected for relatively high flow velocities. If these effects should be taken into account, use the Multiphysics interface [The Multiphase Flow in Porous Media Interface](#) which couples Darcy's Law with a Phase transport equation including capillary pressure and relative permeabilities.

Use the following links to get more information about the Brinkman Equations and the Level Set interfaces and the coupling features.



- [The Brinkman Equations Interface](#)
- [Theory for the Brinkman Equations Interface](#)
- [The Level Set Interface](#) in the *CFD Module User's Guide*
- [Theory for the Level Set Interface](#) in the *CFD Module User's Guide*
- [The Two-Phase Flow, Level Set and Phase Field Interfaces](#) in the *CFD Module User's Guide*
- [Theory for the Two-Phase Flow Interfaces](#) in the *CFD Module User's Guide*.

I n d e x

- A**
 - absolute pressure 46, 178
 - adsorption 223, 234
 - Application Libraries window 22
 - application library examples
 - convection and diffusion 202
 - Darcy's law 102
 - fluid model 271, 272
 - laminar flow 44
 - migration in electric field 202
 - poroelasticity 283
 - Richards' equation 132
 - storage model 110
 - transport of diluted species 196
 - atmosphere/gauge (node) 120
 - average linear velocity 98, 126
- B**
 - Biot-Willis coefficient 275, 284, 290
 - boundary nodes
 - Brinkman equations 173
 - Darcy's law interface 102
 - fracture flow 138
 - free and porous media flow 183
 - transport of diluted species 198
 - boundary stress (node) 56
 - Brinkman equations 167
 - Brinkman equations interface 170
 - theory 167
 - Brooks and Corey retention model 126
- C**
 - CFL number
 - settings 43, 173
 - constitutive relations, poroelasticity 275
 - convection 253
 - convective terms, diluted species 245
 - creeping flow (spf) interface 38
- D**
 - Darcy velocity 116, 253
 - Darcy's law interface 100

- theory 95
 - Darcy's law material model 136
 - Darcy's law, poroelasticity and 278
 - documentation 21
 - domain nodes
 - Brinkman equations 173
 - Darcy's law 102
 - free and porous media flow 183
 - transport of diluted species 198
- E**
 - edge nodes
 - Darcy's law interface 102
 - electrode-electrolyte interface coupling (node)
 - free and porous media flow 113
 - transport of diluted species 216
 - elevation 112, 154
 - emailing COMSOL 22
 - Equilibrium Reaction
 - theory for 243
 - equilibrium reaction (node) 212
- F**
 - Faraday's law 249
 - flow continuity (node) 59
 - fluid and matrix properties (node)
 - fracture flow 140
 - fluid flow
 - Brinkman equations theory 167
 - fracture flow theory 134
 - porous media and subsurface flow 131, 137
 - Richards' equation theory 124
 - flux (node) 236
 - transport of diluted species 236
 - flux discontinuity (node) 208
 - Darcy's law 117
 - transport of diluted species 208
 - fracture flow (node) 123
 - fracture flow interface 137
 - theory 134
 - free and porous media flow interface 181
 - theory 180

- Freundlich exponent 259
- fully developed flow 51
- G** general stress (boundary stress condition) 57
- gravity 61
- H** hydraulic head (node) 120
- Hygroscopic Swelling 226
- hyporheic zones 167
- I** inflow (node) 205
- initial values (node)
 - Brinkman equations 177
 - Darcy's law 110
 - fracture flow 141
 - free and porous media flow 187
 - single-phase, laminar flow 47
 - spf interfaces 47
 - transport of diluted species 202
- inlet (node) 50
 - single-phase flow 50
- internet resources 21
- intrinsic volume averages 167
- K** knowledge base, COMSOL 23
- L** laminar flow interface 39
- Langmuir constant 259
- leaking wall, wall boundary condition 49, 189
- line mass source (node)
 - fluid flow 60
 - species transport 210
- line source
 - species transport 247
- local
 - CFL number 43, 173
- M** magnetostriction (node) 283
- mass based concentrations (node) 202
- mass source (node)
 - Brinkman equations 176
 - Darcy's law 109

- material models, fracture flow 135
- MPH-files 22
- multiphysics
 - magnetostriction 283
 - poroelasticity 282, 287
- N** Nernst-Einstein relation 201, 218, 222, 249
 - no flow (node) 117
 - no flux (node) 204
 - no slip, wall boundary condition 48, 188
 - no viscous stress (open boundary) 56
 - nonconservative formulations 245
 - normal stress, normal flow (boundary stress condition) 57
- O** Ohm's law 250
 - open boundary (node)
 - single-phase flow 56
 - spf interfaces 56
 - transport of diluted species 211
 - outflow (node)
 - transport of diluted species 206
 - outlet (node) 52
 - single-phase flow 52
 - spf interfaces 52
- P** pair nodes
 - Brinkman equations 173
 - Darcy's law interface 102
 - free and porous media flow 183
 - transport of diluted species 198
 - partially saturated porous media (node) 220
 - periodic condition (node)
 - transport of diluted species 209
 - periodic flow condition (node) 58
 - permeability models 107
 - pervious layer (node) 121
 - point mass source (node)
 - fluid flow 60
 - species transport 211
 - point nodes

- Brinkman equations 173
- Darcy's law interface 102
- free and porous media flow 183
- transport of diluted species 198
- point source
 - species transport 246
- poroelasticity interface
 - theory 275
- poroelasticity multiphysics interface 282, 287
- porosity in fracture 140
- porous electrode coupling (node) 215
 - free and porous media flow 113
- porous media and subsurface flow
 - Brinkman equations interface 170
 - Darcy's law interface 100
 - fracture flow interface 137
 - free and porous media flow interface 181
 - Richards' equation interface 131
 - theory, Brinkman equations 167
 - theory, free and porous media flow 180
- porous media and subsurface flow theory 134
- porous media flow modeling 27
- porous media transport properties (node) 216, 220
- porous media, variably saturated 124
- Porous Medium (node)
 - Brinkman equations 174, 185
- pressure (node) 114, 116, 118
- pressure head (node) 120
- pressure point constraint (node) 59
- pseudo time stepping
 - settings 43, 173
- R** Reacting Volume 203, 239
 - reaction coefficients (node) 215
 - reactions (node)
 - transport of diluted species 203
 - retardation factor 259
 - retention models 126
 - Richards' equation interface 131

- theory 124
- S**
 - saturated porous media 124
 - single-phase flow interface
 - laminar flow 39
 - sliding wall 49
 - slip, wall boundary condition 48, 188
 - solids deformation, poroelasticity and 277
 - Stokes equations 38
 - Stokes flow 38
 - storage model 136
 - stratified porous media 257
 - superficial volume average, porous media 167
 - superficial volume averages, porous media 253
 - Supporting Electrolytes 249
 - surface equilibrium reaction (node) 214
 - symmetry (node) 55
 - Darcy's law 117
 - spf interfaces 55
 - transport of diluted species 208
- T**
 - technical support, COMSOL 22
 - theory
 - Brinkman equations 167
 - Darcy's law 95
 - fracture flow 134
 - free and porous media flow 180
 - poroelasticity 275
 - Richards' equation 124
 - transport of diluted species in porous media interface 242
 - transport of diluted species interface 241
 - thickness
 - fracture 229
 - out-of-plane 112
 - thin diffusion barrier (node) 212
 - Thin Impermeable Barrier 212
 - tortuosity factors 255
 - traction boundary conditions 56
 - transport of diluted species in porous media interface 196
 - theory 242

- transport of diluted species interface 192, 231
- theory 241
- V** van Genuchten retention model 126
- volume averages 167
- volume force (node) 47
 - free and porous media flow 187
 - spf interfaces 47
- W** wall (node)
 - free and porous media flow 188
 - single-phase flow 48
- websites, COMSOL 23

