

Pipe Flow Module

User's Guide

Pipe Flow Module User's Guide

© 1998–2025 COMSOL

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

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

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

Version: COMSOL 6.4

Contact Information

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

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

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

Part number: CM022801

C o n t e n t s

Chapter 1: Introduction

About the Pipe Flow Module	10
What Can the Pipe Flow Module Do?	10
The Pipe Flow Module Physics Interface Guide	11
Common Physics Interface and Feature Settings and Nodes	13
The Liquids and Gases Materials Database	14
Where Do I Access the Documentation and Application Libraries?	14
Overview of the User's Guide	17

Chapter 2: Fluid Flow Interfaces

Mechanisms for Modeling Fluid Flow	20
Coupling to Other Physics Interfaces	20
The Pipe Flow Interface	21
Edge, Boundary, Point, and Pair Nodes for the Pipe Flow Interface	22
Fluid Properties	23
Volume Force	25
Pipe Properties	25
Section Orientation	27
Phase Fractions	27
Initial Values	28
Inlet	28
Outlet	30
No Flow	30

Lossless Fitting	30
Pressure	30
Bend	31
Valve	31
Control Valve	32
3-Way Valve	33
Contraction/Expansion	35
Orifice.	35
Pump	36
T-Junction	37
Y-Junction	40
n-Way Junction	41
Internal Pressure Lock.	42
Pipe Flow Physics Feature Symbols	42
The Nonisothermal Pipe Flow Interface	44
Edge, Boundary, Point, and Pair Nodes for the Nonisothermal Pipe Flow Interface.	45
Fluid	46
Heat Transfer	47
The Water Hammer Interface	48
Edge, Boundary, Point, and Pair Nodes for the Water Hammer Interface	50
Fluid Properties	50
Pipe Properties	51
Local Friction Loss	51
Closed.	52
Velocity	52
Pressure	52
Theory for the Pipe Flow Interface	53
Flow Equations	53

Additional Flow Resistances	62
References for the Pipe Flow Interface	71
Theory for the Water Hammer Interface	74
Flow Equations	74
Reference for the Water Hammer Interface	76

Chapter 3: Heat Transfer Interfaces

The Mechanisms for Heat Transfer	78
The Heat Transfer in Pipes Interface	79
Edge, Boundary, Point, and Pair Nodes for the Heat Transfer in Pipes Interface	80
Heat Transfer	80
Heat Source	81
Wall Heat Transfer	82
Internal Film Resistance	83
External Film Resistance	83
Wall Layer	85
Wall Heat Transfer, Inner	86
Initial Values	86
Temperature	86
Heat Outflow	87
Open Pipe End.	87
Theory for the Heat Transfer in Pipes Interface	88
The Heat Transfer Equation.	88
Reference for the Heat Transfer in Pipes Interface	97

Chapter 4: Chemical Species Transport Interfaces

The Transport of Diluted Species in Pipes Interface	100
Edge, Boundary, Point, and Pair Nodes for the Transport of Diluted Species in Pipes Interface	101

Convection and Diffusion	102
Initial Values	102
Junction	103
Reactions.	103
Wall Mass Transfer	104
Concentration	104
Mass Outflow	104
Open Pipe End.	104
The Reacting Pipe Flow Interface	106
Edge, Boundary, Point, and Pair Nodes for the Reacting Pipe Flow Interface.	107
Pipe Properties	108
Initial Values	109
Fluid	109
Fluid Properties	110
Convection and Diffusion	110
Theory for the Transport of Diluted Species in Pipes Interface	112

Chapter 5: Acoustics Interfaces

Chapter 6: Pipe Mechanics

Chapter 7: Multiphysics Couplings

Coupling Features	120
Pipe Connection	120
Pipe Connection, Diluted Species	121
Pipe Wall Heat Transfer	122
Acoustic–Pipe Acoustic Connection	123
Fluid–Pipe Interaction	124

Structure–Pipe Connection 124

Index **125**

Introduction

This guide describes the Pipe Flow Module, an optional add-on package for COMSOL Multiphysics® designed to model and simulate incompressible and weakly compressible flow, heat, and mass transfer in pipes and channels with the Pipe Flow features. Compressible hydraulic transients and acoustics waves can also be modeled using the Water Hammer and Pipe Acoustics features, respectively. To analyze the stresses and deformation in the pipes, the Pipe Mechanics interface is available. The Pipe Flow Module can address problems involving flow velocity, pressure, temperature, stresses, deformation, and sound waves in cooling systems, ventilation systems, pipe systems in the chemical processing industry, and lines in the oil and gas industry.

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

About the Pipe Flow Module

What Can the Pipe Flow Module Do?

The Pipe Flow Module is intended for the modeling and simulation of flow of fluids in the pipes and channel systems, as well as compressible hydraulic transients and acoustics waves. Typical simulations yield the velocity, pressure variation, and temperature in systems of pipes and channels. Hydraulic transients resulting from a valve that is closed rapidly in a pipe network is referred to as water hammer, which can be modeled too. This module has extra functionality for computing displacements, rotations, stresses, and strains in the pipes. The module can be used to design and optimize complex cooling systems in turbines, ventilation systems in buildings, pipe systems in chemical processes, and lines in the oil and gas industry.

In common for pipes and channels that can be modeled using the Pipe Flow Module is that the pipe length is large enough so that the flow inside can be considered fully developed. Piping components such as bends, valves, T-junctions, contractions/expansions, and pumps are also available in the module.

The Pipe Flow Module includes these physics interfaces:

- The *Pipe Flow* interface computes the pressure and velocity field in isothermal pipe systems.
- The *Heat Transfer in Pipes* interface computes the energy balance in pipe systems but receives the flow field as a value or as a known solved field. Wall heat transfer to the surroundings is included.
- The *Transport of Diluted Species in Pipes* interface solves a mass balance equation for pipes in order to compute the concentration distribution of a solute in a dilute solution, considering diffusion, dispersion, convection, and chemical reactions.
- The *Nonisothermal Pipe Flow* interface is a multiphysics interface that solves the flow, pressure, and temperature simultaneously and fully coupled.
- The *Reacting Pipe Flow* interface is a multiphysics interface that solves the flow, pressure, temperature, and reacting species transport simultaneously and fully coupled.
- The *Water Hammer* interface solves rapid hydraulic transients in pipe systems, taking the elastic properties of both the fluid and pipe wall into account.

- The *Pipe Acoustics, Frequency Domain* interface models sound waves in flexible pipe systems, with the assumption of harmonic vibrations.
- The *Pipe Acoustics, Transient* interface models sound waves in flexible pipe systems with arbitrary transient variations in pressure.
- The *Pipe Mechanics* interface analyzes the stresses and deformations in pipes.
- The *Fluid-Pipe Interaction, Fixed Geometry* interface can be used to model problems where the forces from the fluid in a piping system act as structural loads.

The physics interfaces in the module define the conservation of momentum, energy, and mass of a fluid inside a pipe or channel system. The flow, pressure, temperature, and concentration fields across the pipe cross sections are modeled as cross-section averaged quantities, which only vary along the length of the pipes and channels. The pressure losses along the length of a pipe or in a pipe component are described using friction factor expressions. A broad range of built-in expressions for Darcy friction factors cover the entire flow regime from laminar to turbulent flow, Newtonian and non-Newtonian fluids, different cross-sectional geometries, and a wide range of relative surface roughness values. In addition to the continuous frictional pressure drop along pipe stretches, pressure drops due to momentum changes in components such as bends, contractions, expansions, T-junctions and valves are computed through an extensive library of industry standard loss coefficients. Pumps are also available as components.



[Building a COMSOL Multiphysics Model](#) in the *COMSOL Multiphysics Reference Manual*

The Pipe Flow Module Physics Interface Guide

The Pipe Flow Module extends the functionality of the physics interfaces of the base package for COMSOL Multiphysics. The details of the physics interfaces and study types for the Pipe Flow Module are listed in the table. The functionality of the








COMSOL Multiphysics base package is given in the *COMSOL Multiphysics Reference Manual*.



In the *COMSOL Multiphysics Reference Manual*:

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

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
Acoustics				
Pipe Acoustics				
Pipe Acoustics, Frequency Domain		pafd	3D, 2D	eigenfrequency; frequency domain
Pipe Acoustics, Transient		patd	3D, 2D	time dependent
Chemical Species Transport				
Transport of Diluted Species in Pipes		dsp	3D, 2D	stationary; time dependent
Reacting Pipe Flow		rpfl	3D, 2D	stationary; time dependent
Fluid Flow				
Single-Phase Flow				
Pipe Flow		pfl	3D, 2D	stationary; time dependent
Water Hammer		whtd	3D, 2D	time dependent

PHYSICS INTERFACE	ICON	TAG	SPACE DIMENSION	AVAILABLE STUDY TYPE
 Nonisothermal Flow				
Nonisothermal Pipe Flow		nipfl	3D, 2D	stationary; time dependent
Fluid–Pipe Interaction, Fixed Geometry ²		—	3D, 2D	stationary; time dependent
 Heat Transfer				
Heat Transfer in Pipes		htp	3D, 2D	stationary; time dependent
 Structural Mechanics				
Pipe Mechanics		pipem	3D, 2D	stationary; eigenfrequency; time dependent, modal; time dependent, modal reduced-order model; frequency domain; frequency domain, modal; frequency domain, modal reduced-order model; time dependent; response spectrum; random vibration (PSD)

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 Liquids and Gases Materials Database

The Pipe Flow Module includes an additional **Liquids and Gases** material database with temperature-dependent fluid dynamic and thermal properties.



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

Where Do I Access the Documentation and Application Libraries?

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







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




THE DOCUMENTATION AND ONLINE HELP

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




Opening Topic-Based Help

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

-
- | | |
|---|---|
|  | <p>To open the Help window:</p> <ul style="list-style-type: none">• In the Model Builder, Application Builder, or Physics Builder, click a node or window and then press F1.• On any toolbar (for example, Home, Definitions, or Geometry), hover the mouse over a button (for example, Add Physics or Build All) and then press F1.• From the File menu, click Help ().• In the upper-right corner of the COMSOL Desktop, click the Help () button. |
|---|---|
-

-
- | | |
|---|--|
| 
 | <p>To open the Help window:</p> <ul style="list-style-type: none">• In the Model Builder or Physics Builder, click a node or window and then press F1.• In the main toolbar, click the Help () button.• From the main menu, select Help > Help. |
|---|--|
-

Opening the Documentation Window

-
- | | |
|--|---|
| 
 | <p>To open the Documentation window:</p> <ul style="list-style-type: none">• Press Ctrl+F1.• In the main toolbar, click the Documentation () button.• From the main menu, select Help > Documentation. |
|--|---|
-

THE APPLICATION LIBRARIES WINDOW

Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or application. The models and applications are available in COMSOL Multiphysics as MPH files that you can open

for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

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

Opening the Application Libraries Window

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

CONTACTING COMSOL BY EMAIL

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

COMSOL ACCESS AND TECHNICAL SUPPORT

To receive technical support from COMSOL for the COMSOL products, please contact your local COMSOL representative or send your questions to support@comsol.com. An automatic notification and a case number will be sent to you by email. You can also access technical support, software updates, license information, and other resources by registering for a COMSOL Access account.

COMSOL ONLINE RESOURCES

COMSOL website	www.comsol.com
Contact COMSOL	www.comsol.com/contact
COMSOL Access	www.comsol.com/access
Support Center	www.comsol.com/support
Product Download	www.comsol.com/product-download
Product Updates	www.comsol.com/product-update
COMSOL Blog	www.comsol.com/blogs
Discussion Forum	www.comsol.com/forum
Events	www.comsol.com/events
COMSOL Application Gallery	www.comsol.com/models
COMSOL Video Gallery	www.comsol.com/videos
Learning Center	www.comsol.com/support/learning-center
Support Knowledge Base	www.comsol.com/support/knowledgebase

Overview of the User's Guide

The *Pipe 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 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 FLUID FLOW BRANCH

The [Fluid Flow Interfaces](#) chapter describes the Pipe Flow interface and the Nonisothermal Pipe Flow interface including the underlying theory for these physics interfaces.

THE HEAT TRANSFER BRANCH

The [Heat Transfer Interfaces](#) chapter describes the Heat Transfer in Pipes interface and the underlying theory for the physics interface.

THE CHEMICAL SPECIES TRANSPORT BRANCH

The [Chemical Species Transport Interfaces](#) chapter describes the Transport of Diluted Species in Pipes and the Reacting Pipe Flow interfaces and the underlying theory.

THE ACOUSTICS BRANCH

The [Acoustics Interfaces](#) chapter describes the Pipe Acoustics, Transient interface and its theory.


THE STRUCTURAL MECHANICS BRANCH

The [Pipe Mechanics](#) chapter describes the Pipe Mechanics interface.

THE MULTIPHYSICS COUPLINGS

The [Multiphysics Couplings](#) chapter describes the multiphysics couplings available with the Pipe Flow Module. These are the built-in conditions that couple physics together.


Fluid Flow Interfaces



This chapter has information about the physics interfaces in the Pipe Flow Module found under the **Fluid Flow** branch ()



- [Mechanisms for Modeling Fluid Flow](#)
- [The Pipe Flow Interface](#)
- [The Nonisothermal Pipe Flow Interface](#)
- [The Water Hammer Interface](#)
- [Theory for the Pipe Flow Interface](#)
- [Theory for the Water Hammer Interface](#)



See [The Single-Phase Flow, Laminar Flow Interface](#) in the *COMSOL Multiphysics Reference Manual* for other Fluid Flow interface and feature node settings.

Mechanisms for Modeling Fluid Flow

The **Fluid Flow** branch () has a number of subbranches to describe momentum transport. The main focus of this physics area is to model fluids transported in pipe networks in 2D and 3D space, calculating velocity, flow rate, and pressure drops. Pipe flow including heat and mass transfer can also be modeled.

The **Pipe Flow Interface** () , found under the **Single-Phase Flow** branch () when adding a physics interface, is used to model incompressible and weakly compressible fluid flow in piping systems. The equations that are solved are 1D equations that live on 2D boundary segments or 3D edge segments.

The **Nonisothermal Pipe Flow Interface** () , found under the **Nonisothermal Flow** branch () when adding a physics interface, solves a temperature equation together with the equations for compressible flow.



The **Water Hammer Interface** () , found under the **Fluid Flow > Single-Phase Flow** branch () when adding a physics interface, has the equations and boundary conditions for modeling rapid pressure transients in a pipe systems.

Coupling to Other Physics Interfaces

It is often be relevant to couple flow and transport in pipes or pipe networks to physical processes occurring outside the pipe network itself. For instance, liquid can be injected into the surrounding by means of perforated pipes. The Pipe Flow interface can calculate flow rates and pressure drops in a 1D representation in the injection system. The equations can then be coupled to a Fluid Flow interface describing fluid flow in 2D or 3D geometries. Another important coupling that is prepared in the Pipe Flow module is the Wall Heat Transfer multiphysics coupling that couples the temperature in a pipe to the temperature in a 3D surrounding.

More advanced descriptions 2D and 3D continuum fluid flow, such as turbulent and multiphase flow, can be found in the CFD Module. More extensive descriptions of heat transfer, such as in turbulent flow or involving radiation, can be found in the Heat Transfer Module. Furthermore, some applications involving the flow of liquids and gases in porous media are better handled by the Chemical Reaction Engineering Module.

The Pipe Flow Interface

The **Pipe Flow (pfl)** interface () , found under the **Single-Phase Flow** branch () when adding a physics interface, is used to compute the velocity and pressure fields in pipes and channels of different shapes. It approximates the pipe flow profiles flow by 1D assumptions in curve segments, or lines. These lines can be drawn in 2D or 3D and represent simplifications of hollow tubes. The interface is available in 3D on edges, and in 2D on boundaries.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Fluid Properties**, **Pipe Properties**, **Pressure**, **Lossless Fitting**, 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 **Pipe 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 `pf1`.

PHYSICS SYMBOLS

Select the **Enable physics symbols** checkbox to enable the display of physics symbols. See [Pipe Flow Physics Feature Symbols](#) for information about available physics symbols.

Under **Show or hide all physics symbols**, click the **Select All** button, which displays all symbols in that physics interface by selecting all **Show physics symbols** checkboxes in the **Settings** windows for the physics features that include symbols. Similarly, the **Clear All** button clears all **Show physics symbols** checkboxes in the individual physics features.

FLUID MODEL

Select a **Fluid model** — **Newtonian** (the default), **Power Law**, **Bingham**, or **Herschel-Bulkley**.

For **Newtonian** select **Single phase** (the default), **Gas-Liquid**, **friction factor multiplier**, or **Gas-Liquid**, **effective Reynolds number**.

When **Gas-Liquid, friction factor multiplier** or **Gas-Liquid, effective Reynolds number** are selected, the **Phase Fractions** node is automatically added as a default node. Selecting these options also enables the choice of Liquid material and Gas material for the **Fluid Properties** node.

For more information about the fluid models, see the section [Theory for the Pipe Flow Interface](#).

DEPENDENT VARIABLES

This section defines the dependent variables (fields). If required, edit the name, but dependent variables must be unique within a model:

- **Pressure** p (SI unit: Pa)
- **Tangential velocity** u (SI unit: m/s)

DISCRETIZATION

The default discretization for Pipe Flow interface is Linear — that is, piecewise linear interpolation for tangential velocity and pressure.

FURTHER READING



- For an explanation of the tangential velocity, see [Theory for the Pipe Flow Interface](#).
- [Edge, Boundary, Point, and Pair Nodes for the Pipe Flow Interface](#)
- [Theory for the Pipe Flow Interface](#)



- *Discharging Tank*: Application Library path **Pipe_Flow_Module/Tutorials/discharging_tank**
- *Convective Flow in a Heat Exchanger Plate*: Application Library path **Pipe_Flow_Module/Tutorials/heat_exchanger_plate**

Edge, Boundary, Point, and Pair Nodes for the Pipe Flow Interface

The **Pipe Flow Interface** has these 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 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.

- Bend
- Contraction/Expansion
- Control Valve
- Fluid Properties
- Initial Values
- Inlet
- Internal Pressure Lock
- No Flow
- n-Way Junction
- Orifice
- Outlet
- Phase Fractions
- Pipe Properties
- Pressure
- Pump
- T-Junction
- Valve
- Volume Force
- Y-Junction
- Lossless Fitting
- 3-Way Valve



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 is always present and adds the momentum and mass balance equations solved by the physics interface. If volume forces are needed in the model, they should be added by a [Volume Force](#) node. The node also provides settings for defining the material properties of the fluid. If a Liquid-Gas option is selected in the Pipe Flow node settings, properties for both liquid and gas can be entered here.

FLUID PROPERTIES

This subsection displays when **Newtonian > Single phase** is selected as the **Fluid model** for the Pipe Flow interface.

The default **Density** ρ (SI unit: kg/m^3) uses the value **From material**. For **User defined** enter a different value or expression.

The default **Dynamic viscosity** μ (SI unit: Pa·s) uses the value **From material** and describes the relationship between the shear rate and shear stresses in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick (such as oil) have a higher viscosity.

LIQUID PROPERTIES

This section displays when **Newtonian** is selected as the **Fluid model** for the Pipe Flow interface and then **Gas-Liquid, friction factor multiplier** or **Gas-Liquid, effective Reynolds number** are also chosen.

Select the **Liquid material** — **Edge material** (3D components) or **Boundary material** (2D components). If you have more than one material added to the model, you can toggle this setting between the different designated materials.

The default **Density** ρ_L (SI unit: kg/m^3) uses the value **From material**. You can change this to **User defined** to enter a different value or expression.

The default **Dynamic viscosity** μ_L (SI unit: Pa·s) uses the value **From material** and describes the relationship between the shear rate and shear stresses in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick (such as oil) have a higher viscosity.

GAS PROPERTIES

This section displays when **Newtonian** is selected as the **Fluid model** for the Pipe Flow interface and then **Gas-Liquid, friction factor multiplier** or **Gas-Liquid, effective Reynolds number** are also chosen.

Select the **Gas material** — **Edge material** (3D components) or **Boundary material** (2D components). If you have more than one material added to the model, you can toggle this setting between the different designated materials.

The default **Density** ρ_G (SI unit: kg/m^3) uses the value **From material**. For **User defined** enter a different value or expression.

The default **Dynamic viscosity** μ_G (SI unit: Pa·s) uses the value **From material** and describes the relationship between the shear rate and shear stresses in a fluid.

MIXTURE VISCOSITY MODEL

This section displays only when **Newtonian** is selected as the **Fluid model** for the Pipe Flow interface and the **Gas-Liquid, effective Reynolds number** is also chosen.

The default is the **Extended Einstein** which is considered the most accurate method for a large range of gas fractions. The list of options include

- Extended Einstein (default)
- Linear average
- Reciprocal average
- Liquid viscosity
- User defined

For details, please refer to the theory section [Newtonian Fluids](#).

Volume Force

The **Volume Force** node specifies the volume force \mathbf{F} on the right-hand side of the flow equation. Use it, for example, to incorporate the effects of gravity in a model.

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

VOLUME FORCE

Enter the components of the **Volume force \mathbf{F}** (SI unit: N/m^3).

Pipe Properties

The **Pipe Properties** node is used to define the pipe shape, flow resistance, and surface roughness.

For 3D models, a [Section Orientation](#) subnode can be added, in which you specify the orientation of the principal axes of the section. You can add any number of [Section Orientation](#) subnodes if the same section appears with different spatial orientations in the structure.

PIPE SHAPE

Select a pipe shape from the list — **Not set** (the default), **Circular**, **Square**, **Rectangular**, **Annular**, or **User defined**.

- For **Circular** enter a value or expression for the **Inner diameter** d_i (SI unit: m). The default is 10 cm.
- For **Square** enter a value or expression for the **Inner width** w_i (SI unit: m). The default is 5 cm.
- For **Rectangular** enter a value or expression for the **Inner width** w_i (SI unit: m. The default is 5 cm) and **Inner height** h_i (SI unit: m. The default is 10 cm).
- For **annular** enter a value or expression for the **Inner diameter, outer pipe** $d_{i,outer}$ (SI unit: m. The default is 10 cm) and **Outer diameter, inner pipe** $d_{o,inner}$ (SI unit: m. The default is 0 cm).
- For **User defined** enter a value or expression for the **Cross-sectional area** A (SI unit: m^2). The default is $0.01 m^2$. Enter the **Wetted perimeter** Z (SI unit: m). The default is 0.4 m.

FLOW RESISTANCE

Select a **Friction model**.

Newtonian Fluids

Select **Churchill** (the default), **Stokes**, **Wood**, **Haaland**, **Colebrook**, **von Karman**, **Swamee-Jain**, or **User defined**.

- For **Churchill**, **Wood**, **Haaland**, **von Karman**, or **Swamee-Jain** go to the [Surface Roughness](#) section to select from a list of predefined values. Alternatively, enter values or expressions.
- For **User defined** enter a value or expression for the **Darcy friction factor** f_D (dimensionless).

Non-Newtonian Fluids

For Non-Newtonian fluids in tubes of circular cross section the following friction models are available:

For Power-law fluids, the **Irvine**, **Stokes**, and **User defined** options are available.

For Bingham fluids, the **Darby**, and **User defined** options are available.

For Herschel–Bulkley fluids, the **Swamee-Aggarwal**, and **User defined** options are available.

For Non-Newtonian fluids in tubes of noncircular cross section, enter a value or expression for the **Darcy friction factor** f_D (dimensionless).

For more information about the friction models, see [Expressions for the Darcy Friction Factor](#).

SURFACE ROUGHNESS

This section is available if **Churchill, Wood, Haaland, von Karman, or Swamee-Jain** is selected as the **Friction model**.

Select a **Surface roughness** from the list — **Smooth (0 mm)**, **Drawn tubing (0.0015 mm)** (the default), **Glass (0.0015 mm)**, **Thermoplastics (0.0015 mm)**, **Commercial steel (0.046 mm)**, **Wrought iron (0.046 mm)**, **Steel welded seamless (0.061 mm)**, **Asphalted cast iron (0.12 mm)**, **Galvanized iron (0.15 mm)**, **Cast iron (0.26 mm)**, **Wood stave (0.5 mm)**, **Copper and brass (0.61 mm)**, **Concrete (1.5 mm)**, **Riveted steel (4.5 mm)**, or **User defined**.

For **User defined** enter a value or expression for the **Roughness** (SI unit: m).

Section Orientation

Use the **Section Orientation** subnode to define the orientation of a pipe cross section using a reference point or an orientation vector. The default section orientation is always defined. The default orientation can be modified using as many **Section Orientation** subnodes as needed if the same section appears with different spatial orientations in the structure.



This node is available for 3D components.



All settings for the **Section Orientation** subnode are described in the documentation for [Section Orientation](#) in the Pipe Mechanics interface.

Phase Fractions

This node is available when **Newtonian** is selected as the **Fluid model** for the Pipe Flow interface and then **Gas-Liquid, friction factor multiplier** or **Gas-Liquid, effective Reynolds number** are also chosen.

Use the **Phase Fractions** node to define the gas fraction as area averaged void fraction or a quality mass fraction.

GAS FRACTION

Select a **Gas fraction specification** — **Area averaged void fraction (volume fraction)** or **Quality (mass fraction)**.

- For **Area averaged void fraction (volume fraction)** enter an **Area averaged void fraction** ϕ_G (dimensionless). The default is 0.
- For **Quality (mass fraction)** enter a **Quality** ω_G (dimensionless). The default is 0.

Initial Values

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

INITIAL VALUES

Enter values or expressions for the initial value of the **Pressure** p (SI unit: Pa) and the **Tangential Velocity** u (SI unit: m/s).

Inlet

Use the **Inlet** node to set the velocity, volumetric flow rate, or mass flow rate inlet conditions that describe the fluid flow condition at an inlet.



In most cases the inlet boundary conditions appear, some of them slightly modified, in the **Outlet** type as well. This means that there is nothing in the mathematical formulations to prevent a fluid from leaving the domain through boundaries where the **Inlet** type is specified.

INLET SPECIFICATION

Select a **Specification** for the inlet — **Mass flow rate** (the default), **Velocity**, **Volumetric flow rate**. Additionally, **Pump** and **Reservoir** options are available for the Pipe Flow interface.


- For **Mass flow rate**, enter a value or expression for the **Mass flow rate** $q_{m,0}$ (SI unit: kg/s). The default is 0 kg/s.

- For **Velocity**, enter a value or expression for the **Velocity** u_0 (SI unit: m/s). The default is 0 m/s.
- For **Volumetric flow rate**, enter a value or expression for the **Volumetric flow rate** $q_{v,0}$ (SI unit: m³/s). The default is 0 m³/s.
- For **Standard flow, SCCM**, enter a value for standard cubic centimeters per minute. The standard pressure and standard temperature as well as mean molar mass are required as inputs as well. A standard cubic centimeter of gas is the amount of molecules occupying a cubic centimeter at a specified standard temperature and pressure, calculated by the ideal gas law. When used in the oil and gas industry, the standard is often 15 degrees C and 100 kPa, but there are many other slightly different standard conditions used in other circumstances. The inlet velocity calculation is done according to

$$u = \frac{10^{-6} p_0 M}{60 T_0 A \rho R_u} \cdot Q_{\text{SCCM}}$$

- For **Pump**, enter a value for the **External pressure** p_{ext} (SI unit: Pa) and define the relationship between the head and the volumetric flow rate — **Pump curve data**, or **User defined**.

Pump Curve Data

When **Pump curve data** is selected as the **Pump curve**, a table appears, in which the values of the **Flow rate** and the **Head** can be either specified or imported from a text file by clicking the **Load from file** button () under the table. The interpolation between points given in the table is defined using the **Interpolation function type** list in the **Pump Curve Interpolation** section. Units can be specified for the **Flow rate** and the **Head** in the **Units** section.

User Defined

With the **User defined** option, an expression for the **Pump curve** can be specified. The flow rate across the point where this boundary condition is applied is defined by $phys_id.V0$ where $phys_id$ is the name (by default for pipe flow, $phys_id$ is pf1). In order to avoid unexpected behavior, the function used for the pump curve returns the maximum of the user-defined function and 0.

- For **Reservoir**, enter values for reservoir density and pressure. Select a **Entrance type** — **Projecting (K = 1)** (the default), **Sharp edged (K = 0.5)**, **Rounded (K = 0.05)**, or **User-defined loss coefficient**.

Outlet

Use the **Outlet** node to set the velocity, volumetric flow rate, or mass flow rate point conditions that describe the fluid flow condition at an outlet.



See [Inlet](#) for all settings.



All of the formulations for the **Outlet** type are also available, possibly slightly modified, in other boundary types as well. This means that there is nothing in the mathematical formulations to prevent a fluid from entering the domain through boundaries where the **Outlet** boundary type is specified.

No Flow

Use the **No Flow** node to define plugged exits or inlets (no flow).

Lossless Fitting

The **Lossless Fitting** node models junctions without losses. It is a default feature, imposed at all junction points, that will be overridden by subsequent Bend, Valve, Contraction/Expansion, Pump, T-junction, Y-junction, and n -way junction nodes applied at overlapping points.

Pressure

Use the **Pressure** node to define the boundary pressure at points. The pressure condition can only be applied to a point which is connected to exactly one edge (inlet or outlet of a system).

BOUNDARY PRESSURE

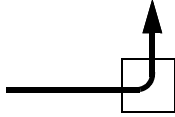
Enter a value or expression for the **Pressure** p_0 (SI unit: Pa). The default is 101325 Pa.

CONSTRAINT SETTINGS

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

Bend

Use the **Bend** node to introduce additional pressure losses due to irreversible turbulent friction in a point associated with a bend in the pipe system.



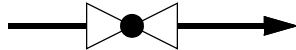
The **Bend** node can only be applied to points that are connected to exactly two edges.

BEND SPECIFICATION

Select a **Bend** — **90 degrees standard elbow (K = 0.9)** (the default), **45 degrees standard elbow (K = 0.5)**, **User-defined loss coefficient**, or **User-defined pressure drop**. For **User-defined loss coefficient** enter a **Loss coefficient K_f** (dimensionless). For **User-defined pressure drop** enter a **Pressure drop** (SI unit: Pa).

Valve

Use the **Valve** node to introduce additional pressure losses due to irreversible turbulent friction in a point associated with the position of a valve in the pipe system. Select a predefined valve type or provide a user-defined loss coefficient.



The **Valve** can only be applied to a point which is connected to exactly two edges.

VALVE SPECIFICATION

Select a **Valve** — **Globe valve (K = 10)** (the default), **Angle valve (K = 4.4)**, **Gate valve (K = 0.2)**, **Ball valve (K = 4.5)**, **Butterfly valve (K = 0.6)**, **Swing check (K = 2.5)**, **User-defined loss coefficient**, or **User-defined pressure drop**. For **User-defined loss coefficient** enter a **Loss coefficient K_f** (dimensionless). For **User-defined pressure drop** enter a **Pressure drop** (SI unit: Pa).

Control Valve

Use the **Control Valve** node to introduce additional pressure losses in order to regulate flow within the system.



The **Control Valve** node can only be applied to points that are connected to exactly two edges.

VALVE SPECIFICATION

The arrow in the graphics window indicates the flow direction. Select the **Reverse direction** checkbox to switch it.

Select a **Specification** for the control valve — **Loss coefficient** (the default), **Flow coefficient**, **Flow factor**, or **Flow rate**.

For **Loss coefficient**, **Flow coefficient**, **Flow factor**, Enter a value or expression for the **Opening fraction** x_{fr} . The default is 1 (valve fully open). Select **Flow characteristic**: Choose how the valve opening affects flow response — Linear, Equal Percentage, or Quick Opening.

Loss Coefficient

Provide two values to define the pressure loss behavior: the pressure loss coefficient for the fully open valve K_{min} and the coefficient at the closed position K_{max} , allowing a small leakage, for **Linear** and **Quick Opening** characteristics. Specify R (Rangeability) instead of K_{max} when the **Equal Percentage** characteristic is selected.

Flow Coefficient

Provide two values that describe the valve capacity to pass fluid: coefficient at full opening Cv_{max} and lower bound Cv_{min} for **Linear** and **Quick Opening** characteristics. Specify R (Rangeability) instead of Cv_{min} when the **Equal Percentage** characteristic is selected.

Flow factor

Provide two values that describe the valve capacity to pass fluid: coefficient at full opening Kv_{max} and lower bound Kv_{min} for **Linear** and **Quick Opening** characteristics. Specify R (Rangeability) instead of Kv_{min} when the **Equal Percentage** characteristic is selected.

Flow Rate

For **Flow rate**, enter a value or expression for the Volumetric flow rate Q_{valve} (SI unit: m^3/s). The default is m^3/s .



- For details see [Control Valve](#) in [Theory for the Pipe Flow Interface](#).

3-Way Valve

Use a 3-way valve to either combine two inlet streams into one outlet or split one stream into two outlets, depending on flow control settings.



The **3-Way Valve** node can only be applied to points that are connected to exactly three edges.

Two **3-Way Valve** nodes cannot be placed on points that are endpoints of the same edge.

POINT SELECTION

Select a junction point to place the 3-way valve.

MAIN BRANCH

Choose the edge where flow is always present as the **Main Branch**. The other edges are assigned as **Side Branch 1** and **Side Branch 2**, which open or close depending on the valve setting.

VALVE SPECIFICATION

The arrow symbols in the graphics window mark **Side Branch 1**. When the opening fraction $x_{\text{fr}} = 1$, flow is fully directed from the **Main Branch** to **Side Branch 1**. Select the **Switch branch** checkbox to change the branch.

Select a **Specification** for the valve — **Loss coefficient** (the default), **Flow coefficient**, or **Flow factor**.

Enter a value or expression for the **Opening fraction** x_{fr} . The default is 1 (**Side Branch 1** is fully open).

SIDE BRANCH 1

Select **Flow characteristic**: Choose how the valve opening affects flow response — Linear, Equal Percentage, or Quick Opening.

Loss Coefficient

Provide two values to define pressure loss behavior: pressure loss coefficient for the fully open valve K_{\min_1} and coefficient at the closed position K_{\max_1} , allowing small leakage, for **Linear** and **Quick Opening** characteristics. Specify R_1 (Rangeability) instead of K_{\max_1} when the **Equal Percentage** characteristic is selected.

Flow Coefficient

Provide two values that describe the valve capacity to pass fluid: coefficient at full opening Cv_{\max_1} and lower bound Cv_{\min_1} for **Linear** and **Quick Opening** characteristics. Specify R_1 (Rangeability) instead of Cv_{\min_1} when the **Equal Percentage** characteristic is selected.

Flow factor

Provide two values that describe the valve capacity to pass fluid: coefficient at full opening Kv_{\max_1} and lower bound Kv_{\min_1} for **Linear** and **Quick Opening** characteristics. Specify R_1 (Rangeability) instead of Kv_{\min_1} when the **Equal Percentage** characteristic is selected.

SIDE BRANCH 2

Select **Flow characteristic**: Choose how the valve opening affects flow response — Linear, Equal Percentage, or Quick Opening.

Loss Coefficient

Provide two values to define pressure loss behavior: pressure loss coefficient for the fully open valve K_{\min_2} and coefficient at the closed position K_{\max_2} , allowing small leakage, for **Linear** and **Quick Opening** characteristics. Specify R_2 (Rangeability) instead of K_{\max_2} when the **Equal Percentage** characteristic is selected.

Flow Coefficient

Provide two values that describe the valve capacity to pass fluid: coefficient at full opening Cv_{\max_2} and lower bound Cv_{\min_2} for **Linear** and **Quick Opening** characteristics. Specify R_2 (Rangeability) instead of Cv_{\min_2} when the **Equal Percentage** characteristic is selected.

Flow factor

Provide two values that describe the valve capacity to pass fluid: coefficient at full opening Kv_{\max_2} and lower bound Kv_{\min_2} for **Linear** and **Quick Opening** characteristics.

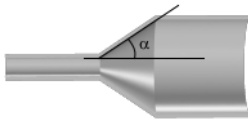
Specify R_2 (Rangeability) instead of $K_{v_{\min_2}}$ when the **Equal Percentage** characteristic is selected.



- For details about pressure loss calculation, see [3-Way Valve](#) in [Theory for the Pipe Flow Interface](#).

Contraction/Expansion

Use the **Contraction/Expansion** node to introduce additional pressure losses due to irreversible turbulent friction in a point associated with the position of a contraction or expansion in the pipe system.



The **Contraction/Expansion** node can only be applied to points that are connected to exactly two pipe segments.

CONTRACTION SPECIFICATION

Select an option from the **Friction** list — **Sudden** (the default), **Gradual**, or **User-defined loss coefficient**.

- For **Gradual**, enter an **Angle of convergence** α (SI unit: radians). The default is 0 radians.
- For **User-defined loss coefficient**, enter a **Loss coefficient** K_f (dimensionless). The default is 0.

Orifice

Use the **Orifice** node to introduce additional pressure losses due to use of orifices in the pipe network.



The **Orifice** node can only be applied to points that are connected to exactly two pipe segments.

ORIFICE SPECIFICATION

Select an option from the **Type** list — **Sharp edged** (the default), or **Round edged**.

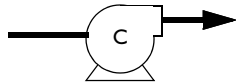
- For **Sharp edged** enter an **Orifice diameter** (SI unit: m).
- For **Round edged**, enter an **Orifice diameter** (SI unit: m) and **Orifice edge radius** (SI unit: m).



- For details about pressure loss calculation, see [Orifice](#) in [Theory for the Pipe Flow Interface](#).
-

Pump

Use the **Pump** node to introduce a pressure jump in a point associated with the position of a pump in the pipe system. Alternatively, specify a fixed mass flow rate for the pump.



The **Pump** can only be applied to a point which is connected to exactly two edges.

PUMP SPECIFICATION


Select a **Pump direction** (available options are based on space dimension) — **Positive X direction** (the default), **Negative X direction**, **Positive Y direction**, **Negative Y direction**, **Positive Z direction**, or **Negative Z direction**. The pressure increases in the direction specified in the **Pump direction**.

Select an option from the **Type** list — **Fixed flow rate** (the default), **Downstream pressure**, **Pressure increase**, and **Pump curve**.

- For **Fixed flow rate**, enter a value or expression for the **Mass flow rate** m_{pump} (SI unit: kg/s). The default is 0 kg/s.
- For **Downstream pressure**, enter a value or expression for the **Pressure** p_{down} (SI unit: Pa). The default is 101,325 Pa.

- For **Pressure increase**, enter a value or expression for the **Pressure increase** Δp (SI unit: Pa). The default is 0 Pa.
- **Pump curve** specifies a relationship between the head and the volumetric flow rate — **Pump curve data** or **User defined**.

Pump Curve Data

When **Pump curve data** is selected as the **Pump curve**, a table appears, where values of the **Flow rate** and the **Head** can be specified or imported from a text file by clicking the **Load from file** button () under the table. The interpolation between points given in the table is defined using the **Interpolation function type** list in the **Pump Curve Interpolation** section. Units can be specified for the **Flow rate** and the **Head** in the **Units** section.

User Defined

With the **User defined** option, an expression for the **Pump curve** can be specified. The flow rate across the point where this boundary condition is applied is defined by $phys_id.V0$ where $phys_id$ is the name (by default for pipe flow, $phys_id$ is pf1). In order to avoid unexpected behavior, the function used for the pump curve returns the maximum of the user-defined function and 0.

T-Junction

Use the **T-Junction** node to specify additional energy losses due to irreversible turbulent friction in a T-junction, which can act as a split or a merger.

SELECTION SETTINGS

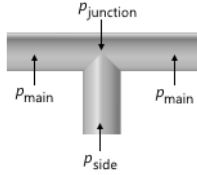


Figure 2-1: A T-junction with local pressure indications.



The **T-junction** is intended for a point that is connected to exactly two collinear main branches and one perpendicular side branch. If this condition is not met, a warning message will be issued when solving. The flow can go in either direction through each of the main branches and the side branch. See [Figure 2-1](#).

The automatic search for **T-junction** points is possible. If you want to use this option, clear the **Manual control of selections** checkbox, the selections will be replaced by the automatic ones. If you want to modify the selection, select **Manual control of selections** again.

FRICTION SPECIFICATION

Several options to specify pressure drop across the junction branches are available.

- The **Loss coefficients** option allows you to enter a value or expression for the **Loss coefficient main branch** K_{main} for the energy loss between main branch and junction (the default is 0.1) and the **Loss coefficient side branch** K_{side} for the energy loss between the side branch and junction (the default is 1.2). Both are dimensionless numbers. See [Figure 2-1](#).
- The **Loss coefficient with respect to common branch** option is available for the Pipe Flow interface. The loss coefficients are calculated automatically. The pressure loss between the branches is expressed in terms of the flow variables in the common branch. For joining flows, the common branch is the collector branch. For separating flow, the common branch is the supplier branch.
- The **Loss coefficients, extended model** option is available for the Pipe Flow interface. This option allows you to specify the loss coefficient in more details and account for

the flow directions. Enter values or expressions for the six dimensionless loss coefficients. See [Figure 2-2](#)-[Figure 2-3](#).

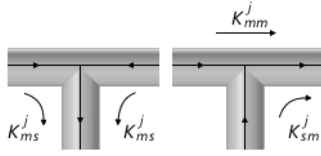


Figure 2-2: T-junction, converging flow.

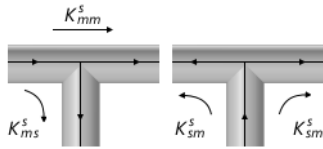


Figure 2-3: T-junction, diverging flow.



- For details about pressure loss calculation, see [T-junctions](#) in [Theory for the Pipe Flow Interface](#).

- Use the **Pressure drops** option to specify a value or expression for the pressure drop explicitly for each branch respectively:

$$\Delta p_{1,s} = p_{\text{main},1} - p_{\text{side}}$$

$$\Delta p_{2,s} = p_{\text{main},2} - p_{\text{side}}$$

Main 1 is the main pipe segment that has the lower index of the two in the selection list (for example, see the list in Pipe Properties).

ADVANCED SETTINGS

The angle tolerance is used to specify the tolerance to a geometry not being a perfect T. Increase the tolerance if you get an error message but accept the geometry even if does not fulfill the requirements for a Y-shape.

CONSTRAINT SETTINGS

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

Y-Junction

Use this feature to specify additional losses due to irreversible turbulent friction in a Y-junction, which can act as a split or a merger. The pressure drop is calculated according to

$$\Delta p = \frac{1}{2}K_f\rho u^2,$$

where K_f is a loss factor. Alternately, specify the pressure drop as a user-defined expression.

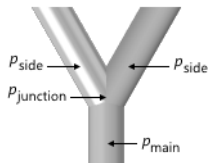


Figure 2-4: A Y-junction with local pressure indications.



The **Y-junction** node can only be applied to points that are connected to exactly two side branches and one main branch. The angle between each of the side branches and main must be equal, and must not be 120 degrees. See [Figure 2-4](#). If these conditions are not met, use an n-way junction. The flow can go in either direction through each of the main branches and the side branch.

The automatic search for **Y-junction** points is possible. If you want to use this option, clear the **Manual control of selections** checkbox, the selections will be replaced by the automatic ones. If you want to modify the selection, select **Manual control of selections** again.

FRICION SPECIFICATION

The **Loss coefficients** options allows you to enter a value or expression for the **Loss coefficient main branch** K_{side} for the energy loss between side and junction and the **Loss coefficient side branch** K_{main} for the energy loss between the main branch and junction (the default is 1.2). Both are dimensionless numbers. See [Figure 2-4](#). The respective pressure losses are calculated as

$$\left| p_{\text{side}, 1} + \frac{1}{2} \rho_{\text{side}, 1} u_{\text{side}, 1}^2 - p_{\text{junction}} \right| = \frac{1}{2} K_{f, \text{side}} \rho_{\text{side}, 1} u_{\text{side}, 1}^2$$

$$\left| p_{\text{side}, 2} + \frac{1}{2} \rho_{\text{side}, 2} u_{\text{side}, 2}^2 - p_{\text{junction}} \right| = \frac{1}{2} K_{f, \text{side}} \rho_{\text{side}, 2} u_{\text{side}, 2}^2$$

$$\left| p_{\text{main}} + \frac{1}{2} \rho_{\text{main}} u_{\text{main}}^2 - p_{\text{junction}} \right| = \frac{1}{2} K_{f, \text{main}} \rho_{\text{main}} u_{\text{main}}^2 .$$

For Nonisothermal Pipe Flow and Reacting Pipe Flow interface interfaces, the respective pressure drops are calculated as

$$|p_{\text{side}, 1} - p_{\text{junction}}| = \frac{1}{2} K_{f, \text{side}} \rho u_{\text{side}, 1}^2$$

$$|p_{\text{side}, 2} - p_{\text{junction}}| = \frac{1}{2} K_{f, \text{side}} \rho u_{\text{side}, 2}^2$$

$$|p_{\text{main}} - p_{\text{junction}}| = \frac{1}{2} K_{f, \text{main}} \rho u_{\text{main}}^2 .$$

The advantage of using loss coefficients is that they are readily available in engineering tables and textbooks.

Use the **Pressure drops** option to specify a value or expression for the pressure drop explicitly for each branch respectively:

$$\Delta p_{1, m} = |p_{\text{main}} - p_{\text{side}, 1}|$$

$$\Delta p_{2, m} = |p_{\text{main}} - p_{\text{side}, 2}|$$

Side 1 is the main pipe segment that has the lower index of the two in the selection list (for example, see the list in Pipe Properties).

ADVANCED SETTINGS

The angle tolerance is used to specify the tolerance to a geometry not being a perfect Y. Increase the tolerance if you get an error message but accept the geometry even if does not fulfill the requirements for a Y-shape.

n-Way Junction

Use the **n-Way Junction** feature to specify additional pressure losses due to irreversible turbulent friction in an n-way junction that does not fall into the categories Y- or

T-junctions. The number of pipe connections to a point can be any number, and the angles between the connection pipes arbitrary.

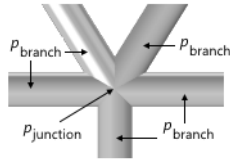


Figure 2-5: An n -Way Junction with local pressure indications.

FRICION SPECIFICATION

The **Loss coefficients** options allows you to enter a value or expression for the **Loss coefficient main branch** K_{branch} for the energy loss between branch and junction.

The pressure loss is calculated as

$$\left| p_{\text{branch}, i} + \frac{1}{2} \rho_{\text{branch}, i} u_{\text{branch}, i}^2 - p_{\text{junction}} \right| = \frac{1}{2} K_{\text{branch}} \rho_{\text{branch}, i} u_{\text{branch}, i}^2$$

For Nonisothermal Pipe Flow and Reacting Pipe Flow interfaces, the pressure drop is calculated as

$$|p_{\text{branch}, i} - p_{\text{junction}}| = \frac{1}{2} K_{\text{branch}} \rho u_{\text{branch}, i}^2$$

Internal Pressure Lock











Use the **Internal Pressure Lock** node to specify the pressure at a point. The **Internal Pressure Lock** condition can only be applied to a point which connects exactly two edges.

Pipe Flow Physics Feature Symbols



For pipe flow interface, the physics symbols are available for internal points boundary conditions. To display the boundary condition symbols listed in [Table 2-1](#), enable the **Show physics symbols** in the Physics Symbols section in the Settings window for the main **Pipe Flow** node. The checkbox is not selected by default.

These symbols are available with the applicable pipe flow feature nodes in the Pipe Flow interface are reproduced in the table below:

TABLE 2-1: PIPE FLOW BOUNDARY CONDITION PHYSICS SYMBOLS.

SYMBOL	SYMBOL NAME	DISPLAYED BY NODE
	Bend	Bend
	Valve	Valve
	Pump	Pump
	Contraction/Expansion	Contraction/Expansion
	T-Junction	T-Junction
	Y-Junction	Y-Junction
	n-Way Junction	n-Way Junction
	Orifice	Orifice
	Control Valve	Control Valve
	3-Way Valve	3-Way Valve

The Nonisothermal Pipe Flow Interface

The **Nonisothermal Pipe Flow (nipfl)** interface () found under the **Nonisothermal Flow** branch () when adding a physics interface, is used to compute the temperature, velocity, and pressure fields in pipes and channels of different shapes. It approximates the pipe flow profile by 1D assumptions in curve segments, or lines. These lines can be drawn in 2D or 3D and represent simplifications of hollow tubes. The physics interface is available in 3D on edges and 2D on boundaries.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Fluid**, **Pipe Properties**, **Pressure**, **Temperature**, 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 **Nonisothermal Pipe 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 `nipfl`.

FLUID MODEL



Select a **Fluid model** — **Newtonian** (the default), **Power law**, or **Bingham**.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields). If required, edit the name, but dependent variables must be unique within a model:

- **Pressure** p (SI unit: Pa)
- **Tangential velocity** u (SI unit: m/s)
- **Temperature** T (SI unit: K)

FURTHER READING

	<ul style="list-style-type: none">• For an explanation of the tangential velocity, see Theory for the Pipe Flow Interface.• Edge, Boundary, Point, and Pair Nodes for the Nonisothermal Pipe Flow Interface• Theory for the Heat Transfer in Pipes Interface
	<p><i>Insulation of a Pipeline Section:</i> Application Library path Pipe_Flow_Module/Heat_Transfer/pipeline_insulation</p>

Edge, Boundary, Point, and Pair Nodes for the Nonisothermal Pipe Flow Interface

Because [The Nonisothermal Pipe Flow Interface](#) is a multiphysics interface it shares many nodes with other physics interfaces. These boundary, edge, point, and pair nodes are available as indicated (and listed in alphabetical order).

Described in this section:

- [Fluid](#)
- [Heat Transfer](#)

These nodes are described for the Reacting Pipe Flow interface:

- [Fluid Properties](#)
- [Pipe Properties](#)

These nodes are described for the Heat Transfer in Pipes interface:

- [External Film Resistance](#)
- [Internal Film Resistance](#)
- [Wall Heat Transfer](#)
- [Wall Layer](#)

These nodes are described for the Heat Transfer interface in the *COMSOL Multiphysics Reference Manual*:

- Heat Source
- Outflow
- Temperature

These nodes are described for the Pipe Flow interface (listed in alphabetical order):

- | | |
|-------------------------|----------------|
| • Bend | • Pressure |
| • Contraction/Expansion | • Pump |
| • Inlet | • T-Junction |
| • No Flow | • Valve |
| • n-Way Junction | • Volume Force |
| • Outlet | • Y-Junction |
| • Phase Fractions | |



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

Use the **Fluid** node to define the density, dynamic viscosity, and heat convection and conduction properties.

PRESSURE WORK

The **Pressure Work** section has an **Include pressure work** checkbox where a pressure work term can be activated. When checked a contribution to the right-hand side of the Heat Transfer in Pipes equation is added.

This term can be important to activate if the pressure drop is expected to be considerable and the fluid is compressible. The pressure-volume work is added to the energy balance.

For more information, see the section [Theory for the Heat Transfer in Pipes Interface](#).

FLUID PROPERTIES

The default **Density** ρ and **Dynamic viscosity** is specified here.

HEAT CONVECTION AND CONDUCTION

The default **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)), **Ratio of specific heats** γ (dimensionless), and **Thermal conductivity** k (SI unit: W/(m·K)) all use values **From material**. For **User defined** enter different values or expressions.

Heat Transfer

Use the **Heat Transfer** node to define the density, yield stress, plastic viscosity, and heat convection and conduction properties.

FLUID PROPERTIES



The default **Density** ρ (SI unit: kg/m³) is **User defined** and is $1 \cdot 10^3$ kg/m³.

Enter a value or expression for the **Yield stress** τ_s (SI unit: Pa) and **Plastic viscosity** μ_s (SI unit: Pa·s). The defaults are 0 Pa and 0 Pa·s, respectively.

HEAT CONVECTION AND CONDUCTION

The default **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)), **Ratio of specific heats** γ (dimensionless), and **Thermal conductivity** k (SI unit: W/(m·K)) all use values **From material**. For **User defined** enter different values or expressions.

The Water Hammer Interface

The **Water Hammer (whtd)** interface () , found under the **Fluid Flow > Single-Phase Flow** branch () when adding a physics interface, is used to compute the velocity and pressure fields for propagation of a hydraulic transient, also known as a water hammer, in elastic pipe systems. For example, the propagation of a water hammer in a piping system of an oil refining plant.

The equations governing the propagation of hydraulic transients in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. In the formulation of the Water Hammer interface, the compressibility effects of the fluid and pipe walls are linearized. The resulting equations are expressed in the cross-sectional averaged variables and reduce the equations to a 1D component with scalar field variables. The physics interface is available in 3D on edges and at points, and in 2D on boundaries and points.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Fluid Properties**, **Pipe Properties**, **Closed**, and **Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Water Hammer** 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 `whtd`.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields). If required, edit the name, but dependent variables must be unique within a model:

- **Pressure** p (SI unit: Pa)
- **Tangential velocity** u (SI unit: m/s)



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.



- [Edge, Boundary, Point, and Pair Nodes for the Water Hammer Interface](#)
 - [Theory for the Pipe Flow Interface](#)
-



Water Hammer: Application Library path **Pipe_Flow_Module/Verification_Examples/water_hammer_verification**

Edge, Boundary, Point, and Pair Nodes for the Water Hammer Interface

The **Water Hammer Interface** has these edge, boundary, point, and pair nodes available and listed in alphabetical order:

- [Closed](#)
- [Fluid Properties](#)
- [Local Friction Loss](#)
- [Pipe Properties](#)
- [Pressure](#)
- [Velocity](#)
- The [Initial Values](#) and [Volume Force](#) nodes are described for the Pipe Flow interface.



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 equations solved by the physics interface, except for volume forces which are added by the [Volume Force](#) node. The node also provides an interface for defining the physical properties of the fluid.

PHYSICAL PROPERTIES

Select a **Fluid model** — **Linear elastic** (the default).

The default **Density** ρ (SI unit: kg/m^3) uses the value **From material**. For **User defined** enter a different value or expression.

The default **Dynamic viscosity** μ (SI unit: Pa·s) uses the value **From material** and describes the relationship between the shear rate and the shear stresses in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick (such as oil) have a higher viscosity.

The default **Speed of sound** c_s (SI unit: m/s) uses the value **From material**. For **User defined** enter a different value or expression.

Pipe Properties

The **Pipe Properties** node is used to define the pipe shape, pipe model, and flow resistance.

PIPE SHAPE

The **Pipe Shape** settings are the same as for the **Pipe Properties** node described for the Pipe Flow interface.

PIPE MODEL

Select a **Pipe model** — **Zero axial stress** (the default), **Anchored at one end**, **Anchored at both ends**, or **Incompressible cross section**.

- For **Zero axial stress**, select a **Young's modulus E** — **Not set** (the default) or **User defined**. If **User defined** is selected, enter a value or expression.
- For **Anchored at one end** or **Anchored at both ends**, enter a value or expression for **Young's modulus E** . Select **Poisson's ratio ν** — **Not set** (the default) or **User defined**. For **User defined** enter a value or expression.

When **Zero axial stress**, **Anchored at one end**, or **Anchored at both ends** is selected, also select a **Wall thickness Δw** — **Not set** (the default) or **User defined**. For **User defined** enter a value or expression.

FLOW RESISTANCE

The **Flow Resistance** settings are the same as for the **Pipe Properties** node described for the Pipe Flow interface.

Local Friction Loss

Use the **Local Friction Loss** node to add a lumped friction loss that depends on the velocity according to [Equation 2-44](#).

FRICTION SPECIFICATION

Enter a **Loss coefficient K_f** (dimensionless). The default is 0.

CONSTRAINT SETTINGS

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

Closed

Use the **Closed** node to impose zero velocity.

Velocity

Use the **Velocity** node to prescribe a velocity.

VELOCITY

Enter a value or expression for u_{in} (SI unit: m/s). The default is 0 m/s.

Pressure

Use the **Pressure** node to define the boundary pressure at points.

PRESSURE

Enter a value or expression for the **Pressure** p (SI unit: Pa).

CONSTRAINT SETTINGS

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

Theory for the Pipe Flow Interface

The [Pipe Flow Interface](#) and [The Nonisothermal Pipe Flow Interface](#) theory is described in this section:

- [Flow Equations](#)
- [References for the Pipe Flow Interface](#)

Flow Equations

The [Pipe Flow Interface](#) calculates the pressure and velocity of an incompressible or weakly compressible fluid by solving the continuity and momentum equations outlined below.

A one dimensional pipe can be present on a boundary in a 2D geometry, or on an edge in a 3D geometry.

MOMENTUM AND CONTINUITY EQUATIONS

The momentum and continuity equations for flow in a pipe are given by ([Ref. 17](#)):

$$\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p - f_D \frac{\rho}{2d_h} \mathbf{u} |\mathbf{u}| + \mathbf{F} \quad (2-1)$$

and

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

The second term on the right-hand side in [Equation 2-1](#) represents the pressure drop due to viscous shear. Here, \mathbf{u} is the cross-section averaged velocity (SI unit: m/s), ρ the density (SI unit: kg/m³), p pressure (SI unit: Pa), f_D (dimensionless) the Darcy friction factor (see [Expressions for the Darcy Friction Factor](#)) and \mathbf{F} is a volume force term (SI unit: N/m³).

Furthermore, d_h is the mean hydraulic diameter (SI unit: m), given by:

$$d_h = \frac{4A}{Z} \quad (2-3)$$

where A is the pipe cross section area (SI unit: m^2) available for flow, and Z is the wetted perimeter (SI unit: m).



Figure 2-6: Unit tangent vector to the pipe axis.

Let

$$\mathbf{e}_t = \begin{pmatrix} e_{t,x} & e_{t,y} & e_{t,z} \end{pmatrix}$$

be the unit tangent vector to the pipe axis.

If the following assumptions apply:

- The velocity profile is fully developed with an entire pipe section.
- The cross section area is allowed to change between pipe segments and within pipe segments.
- Empirical functions (friction charts) describe viscous pressure drop both for turbulent and laminar flow regimes
- The curvature of the pipe segment gives rise to insignificant pressure loss in comparison with wall friction.
- Shocks are neglected
- All velocity components normal to the pipe axis is assumed to be 0,

the momentum balance in [Equation 2-1](#) can be rewritten as

$$\mathbf{e}_t \cdot \left[\rho \frac{\partial \mathbf{u}}{\partial t} + \rho \mathbf{u} \cdot \nabla \mathbf{u} = -\nabla p - f_D \frac{\rho}{2d_h} \mathbf{u} |\mathbf{u}| + \mathbf{F} \right] \quad (2-4)$$

and we can define the *tangential velocity* u as $\mathbf{u} = u \mathbf{e}_t$. The pipe flow physics interface in COMSOL Multiphysics solves for the tangential velocity u . This is also the quantity the user specifies in for example inflow boundary conditions and in the initial conditions settings.

Curvature pressure drop effects can be added in points with the [Bend](#) feature.

Gravity can be included explicitly in the model, but when the variation in density is negligible, and the model is not pressure driven (the inlet boundary condition is a pressure setting), the only effect of including gravity is a change in the total pressure

level. In those cases, it is therefore common modeling practice to set the gravity \mathbf{F} to 0 and interpret the pressure variable as the reduced pressure $p = p - \rho g(z_0 - z)$, where z_0 is the datum level of the free liquid surface. This reduces the model complexity and yields the same results.

The Darcy friction factor in Equation 2-1 accounts for the continuous pressure drop along a pipe segment due to viscous shear, and is expressed as a function of the Reynolds number (Re) and the surface roughness divided by the hydraulic diameter (e/d_h).

$$f_D = f_D\left(\text{Re}, \frac{e}{d_h}\right) \quad (2-5)$$

where

$$\text{Re} = \frac{\rho u d_h}{\mu} \quad (2-6)$$

The physics interface automatically calculates f_D from one of the predefined expressions Equation 2-7 through Equation 2-13.

Expressions for the Darcy Friction Factor

Newtonian Fluids For single-phase fluids, the Churchill equation (Ref. 1) for the Darcy friction factor can be used for the full range of Re (laminar, transition, and turbulent) and full range of e/d :

$$f_D = 8 \left[\left(\frac{8}{\text{Re}} \right)^{12} + (A + B)^{-1.5} \right]^{1/12} \quad (2-7)$$

where

$$A = \left[-2.457 \ln \left(\left(\frac{7}{\text{Re}} \right)^{0.9} + 0.27(e/d) \right) \right]^{16} \quad (2-8)$$

$$B = \left(\frac{37530}{\text{Re}} \right)^{16} \quad (2-9)$$

In the laminar regime ($\text{Re} < 2000$), f_D is independent of the surface roughness and is given by the Stokes formula:

$$f_D = \frac{64}{\text{Re}} \quad (2-10)$$

The Wood equation (Ref. 2) gives the friction factor for $4000 < \text{Re} < 1 \cdot 10^7$ and $1 \cdot 10^{-5} < e/d < 0.04$, according to

$$f_D = 0.094(e/d)^{0.225} + 0.53(e/d) + 88(e/d)^{0.44} \cdot \text{Re}^a \quad (2-11)$$

with

$$a = -1.62(e/d)^{0.134} \quad (2-12)$$

The Haaland equation (Ref. 3) for the Darcy friction factor is commonly used for oil pipelines and wells. It can recover both small and large relative roughness limits for a wide range of Reynolds numbers ($4 \cdot 10^3 < \text{Re} < 1 \cdot 10^8$)

$$\sqrt{\frac{1}{f_D}} = -1.8 \log_{10} \left(\left(\frac{e/d}{3.7} \right)^{1.11} + \left(\frac{6.9}{\text{Re}} \right) \right) \quad (2-13)$$

It can be rewritten as

$$f_D = \frac{(1/1.8)^2}{\left(\log_{10} \left(\frac{e/d}{3.7} \right)^{1.11} + \left(\frac{6.9}{\text{Re}} \right) \right)^2} \quad (2-14)$$

For very low relative roughness e/d , the Haaland equation simplifies to Colebrook's explicit formula (Ref. 4)

$$\sqrt{\frac{1}{f_D}} = -1.8 \log_{10} \left(\frac{6.9}{\text{Re}} \right) \quad (2-15)$$

For very large relative roughness e/d , the Haaland equation simplifies to the von Kármán formula (Ref. 5)

$$\sqrt{\frac{1}{f_D}} = -1.8 \log_{10} \left(\left(\frac{e/d}{3.7} \right)^{1.11} \right) = -2 \log_{10} \left(\frac{e/d}{3.7} \right) \quad (2-16)$$

An alternative to Haaland equation is the Swamee–Jain equation (Ref. 6)

$$f_D = \frac{0.25}{\left(\log_{10} \left(\left(\frac{e/d}{3.7} \right) + \frac{5.74}{(\text{Re})^{0.9}} \right) \right)^2} \quad (2-17)$$

The equation is valid for relative roughness $1 \cdot 10^{-6} < e/d < 1 \cdot 10^{-2}$ and for Reynolds number in the range $5 \cdot 10^3 < \text{Re} < 1 \cdot 10^8$.

All the above equations are selectable from a list of friction factor expressions. As noted, only the Churchill equation covers both the laminar and turbulent flows, as well as the transitional region in between these flow regimes. The equations by Wood, Haaland, Colebrook, von Kármán, or Swamee–Jain, intended for the turbulent regime, are combined with the Stokes equation for laminar flow to cover all flow conditions. When $Re < 1000$, COMSOL Multiphysics selects Stokes equation if it predicts a friction factor greater than equations for the turbulent regimes. This produces more accurate results than using the turbulent friction factors in the laminar regime, but it does not necessarily produce accurate estimates of the friction factor in the transition region. Therefore, it is always advisable to check the Reynolds number for a given pipe flow solution and change the friction model and recalculate, if necessary.

The Newtonian fluid type also has two *Gas-Liquid* options, which employ a simple two-phase approach presented by Balasubramaniam and others (Ref. 26). This is a one-fluid approach that treats the fluid as one phase (one mass and momentum balance is solved), but it corrects the pressure drop correlation with empirical factors for liquid-gas mixtures. As such, the model also makes the following assumptions:

- The velocity of the gas and liquid phases are the same, meaning that no liquid holdup is accounted for.
- No phase separation occurs due to gravity or other effects.
- No phase transition (condensation or vaporization) is assumed.
- The gas phase-fraction is assumed to be small so that the mixture is virtually incompressible.

The first selectable method is the *gas-liquid, friction factor multiplier*, which modifies the single-phase Newtonian Darcy friction factor defined in Equation 2-7 and on, such that

$$f_D = \Phi_{2P}^2 \cdot f_{D,L} \quad (2-18)$$

where $f_{D,L}$ is the Newtonian single phase friction factor computed as described earlier in this section. f_D is then used as usual in the pressure drop correlation in Equation 2-4. The two-phase friction factor Φ_{2P}^2 is calculated as (Ref. 26)

$$\Phi_{2P}^2 = \frac{\rho_L}{\rho} \left[1 + \omega_G \frac{(\rho_L - \rho_G)}{\rho_G} \right] \left[1 + \omega_G \frac{(\mu_L - \mu_G)}{\mu_G} \right]^{-1/4} \quad (2-19)$$

where ρ_L , μ_L , ρ_G , and μ_G are the density and viscosity of the liquid, and gas, respectively, and where ω_G is the quality (the gas phase mass fraction). The total density in [Equation 2-19](#) is calculated as

$$\rho = \phi_G \rho_G + (1 - \phi_G) \rho_L \quad (2-20)$$

or

$$\rho = \frac{1}{\frac{\omega_G}{\rho_G} + \frac{(1 - \omega_G)}{\rho_L}} \quad (2-21)$$

where ϕ_G is the user provided volumetric void fraction. The user has a choice to either enter gas phase mass fraction ω_G or volumetric void fraction ϕ_G . The conversion formula between the two is

$$\phi_G = \frac{\rho}{\rho_G} \omega_G \quad (2-22)$$

The second selectable gas-liquid method is the *gas-liquid, effective Reynolds number*. This method uses an effective adjusted viscosity to calculate the Reynolds number in the pressure loss calculations. The default mixture viscosity model is the extended Einstein, which probably is the most accurate within a large range of gas mass fractions ([Ref. 27](#)):

$$\mu = \mu_L (1 - \phi_G)^{-(\mu_L + 2.5\mu_G)/(\mu_L + \mu_G)} \quad (2-23)$$

and the other options are:

- Linear average

$$\mu = \mu_L (1 - \phi_G) + \mu_G \phi_G \quad (2-24)$$

- Reciprocal average

$$\frac{1}{\mu} = \frac{(1 - \phi_G)}{\mu_L} + \frac{\phi_G}{\mu_G} \quad (2-25)$$

- Liquid viscosity

$$\mu = \mu_L \quad (2-26)$$

- User defined

Non-Newtonian Power-law Fluids For Power-law fluids the apparent viscosity is related to the shear rate as

$$\mu_{\text{app}} = m_p (\dot{\gamma})^{n-1}$$

where m_p and n are two empirical curve fitting parameters known as the fluid consistency coefficient and the flow behavior index, respectively. Note that the dimensions of the consistency coefficient, m_p , depend on value of the value of n . The power law relation can be written as

$$\mu_{\text{app}} = m \left(\frac{\dot{\gamma}}{\dot{\gamma}_{\text{ref}}} \right)^{n-1} \quad (2-27)$$

Here $m = m_p (\dot{\gamma}_{\text{ref}})^{n-1}$ is the value of apparent viscosity at the reference share rate $\dot{\gamma}_{\text{ref}}$. The reference shear rate $\dot{\gamma}_{\text{ref}}$ has the default value 1 s^{-1} . In the laminar regime the friction factor for power law fluids can be calculated by the Stokes equation using the modified Reynolds number proposed by Metzner and Reed (Ref. 7):

$$f_D = \frac{64}{\text{Re}_{\text{MR}}} \quad (2-28)$$

$$\text{Re}_{\text{MR}} = \frac{\rho u^{(2-n)} d_h^n}{8^{(n-1)} m_p \left(\frac{3n+1}{4n} \right)^n} \quad (2-29)$$

For the turbulent regime Irvine (Ref. 8) proposed the following expression for the friction factor:

$$f_D = 4 \left(\frac{D(n)}{\text{Re}_{\text{MR}}} \right)^{\left(\frac{1}{3n+1} \right)} \quad (2-30)$$

where

$$D(n) = \frac{2^{(n+4)}}{7^{7n}} \left(\frac{4n}{3n+1} \right)^{3n^2} \quad (2-31)$$

Ryan and Johnson (Ref. 9) formulated a criterion for the transition between laminar and turbulent flow, where

$$\text{Re}_{\text{MR},c} = \frac{6464n}{(3n+1)^2} (2+n)^{\frac{(2+n)}{1+n}} \quad (2-32)$$

predicts the critical Reynolds number.

Non-Newtonian Bingham Plastic Model. The Bingham model describes viscoplastic fluid with a yield stress:

$$\begin{aligned} \tau &= \tau_B + \mu_B \dot{\gamma} \quad \text{for } |\tau| > |\tau_B| \\ \dot{\gamma} &= 0 \quad \text{for } |\tau| < |\tau_B| \end{aligned} \quad (2-33)$$

The yield stress, τ_B (SI unit: Pa), and the plastic viscosity, μ_B (SI unit: Pa-s), are found by curve fitting to experimental data.

The Swamee-Aggarwal equation (Ref. 10) gives the friction factor for a Bingham plastic fluid in the laminar regime according to:

$$f_{\text{D}, \text{laminar}} = \frac{64}{\text{Re}_B} + \frac{10.67 + 0.1414 \left(\frac{\text{He}}{\text{Re}_B} \right)^{1.143}}{\left(1 + 0.149 \left(\frac{\text{He}}{\text{Re}_B} \right)^{1.16} \right) \text{Re}_B} \left(\frac{\text{He}}{\text{Re}_B} \right) \quad (2-34)$$

where

$$\text{Re}_B = \frac{\rho u d_h}{\mu_B} \quad (2-35)$$

and

$$\text{He} = \frac{\rho d_h^2 \tau_B}{\mu_B^2} \quad (2-36)$$

is the Hedström number.

For turbulent flow Darby (Ref. 11) provided the equation:

$$f_{\text{D}, \text{turbulent}} = 4 \cdot 10^{a_0} \text{Re}_B^{-0.193} \quad (2-37)$$

with

$$\alpha_0 = -1.47(1 + \exp(-2.9 \cdot 10^{-5} \text{He})) \quad (2-38)$$

as well as an equation covering all flow regimes:

$$f_D = (f_{D, \text{laminar}}^b + f_{D, \text{turbulent}}^b)^{\frac{1}{b}} \quad (2-39)$$

where

$$b = 1.7 + \frac{40000}{\text{Re}_B} \quad (2-40)$$

Non-Newtonian Herschel–Bulkley Model. The Herschel–Bulkley non-Newtonian model is a generalization of Bingham plastic model

$$\begin{aligned} \tau &= \tau_B + m \left(\frac{\dot{\gamma}}{\dot{\gamma}_{\text{ref}}} \right)^{n-1} \dot{\gamma} \quad \text{for } |\tau| > |\tau_B| \\ \dot{\gamma} &= 0 \quad \text{for } |\tau| < |\tau_B| \end{aligned} \quad (2-41)$$

The yield stress, τ_B (SI unit: Pa), and the consistency coefficient, m (SI unit: Pa·s), and flow behavior index, n are found by curve fitting to experimental data.

The Swamee–Aggarwal equation (Ref. 12) gives the friction factor for a Herschel–Bulkley fluid in the laminar regime according to:

$$f_{D, \text{laminar}} = \frac{64}{\text{Re}_{\text{MR}}} + \frac{64}{\text{Re}_{\text{MR}}} \left[\frac{1}{\left(36 + \left[\frac{1.5}{n} \right]^{2.46} \right)^{0.5}} \frac{\text{He}}{\text{Re}_{\text{MR}}} \right]^{\frac{0.958n}{2-n}} \quad (2-42)$$

where Re_{MR} is the modified Reynolds number given by Equation 2-29 and

$$\text{He} = \frac{\rho d_h^2}{m_p} \left(\frac{\tau_B}{m_p} \right)^{\frac{2-n}{n}} \quad (2-43)$$

is the Hedström number.



The non-Newtonian friction models outlined above do not include any effects of wall roughness. Since the laminar sublayers tend to be thicker than non-Newtonian than for Newtonian fluids, the effect of pipe roughness is likely to be smaller. Friction models including the surface roughness can be used as user-defined specifications and can be found in for example (Ref. 13).

The non-Newtonian friction models outlined above apply for pipes with circular cross section. The generalized Reynolds number needs to be modified to account for other cross sections.

Surface Roughness

Values of the absolute surface roughness found in the literature (Ref. 14 and Ref. 15) are reproduced in the table below:

TABLE 2-2: SURFACE ROUGHNESS.

TUBE MATERIAL	SURFACE ROUGHNESS (MM)
Smooth	0
Drawn tubing	0.0015
Glass	0.0015
Thermoplastics	0.0015
Commercial steel and wrought iron	0.046
Steel, welded seamless	0.061
Asphalted cast iron	0.12
Galvanized iron	0.15
Cast-iron	0.26
Wood stave	0.18–0.9
Concrete	0.3–3
Riveted steel	0.9–9
Copper and brass	0.61

Additional Flow Resistances

Additional flow resistance due to irreversible turbulent effects in joints and junctions can be added as point conditions between pipe segments — Bends, Valves, Orifices,

T-junctions, Y-junctions, n-way junctions, Contractions, and Expansions. These resistances give rise to abrupt (lumped) energy losses due to turbulence in the points where they are added. The lumped pressure drop is calculated as:

$$\Delta p = \frac{1}{2}K_f \rho u^2 \quad (2-44)$$

For more information about each type, see the interface descriptions in [The Pipe Flow Interface](#) section.

Loss coefficients, K_i , for turbulent flow are available in the literature ([Ref. 15](#)) and the set predefined in the Pipe Flow interface is reproduced in the table below:

TABLE 2-3: LOSS COEFFICIENTS.

FEATURE	DESCRIPTION	LOSS COEFFICIENT K
Bends	90° standard elbow	0.9
	45° standard elbow	0.5
Valves	Globe, fully open	10
	Angle, fully open	4.4
	Gate, fully open	0.2
	Ball, fully open	4.5
	Butterfly, fully open	0.6
	Swing check, fully open	2.5
T-junction	T-junction split/merger	0.1 and 2.1 for side and main. See Figure 2-1 for a description of flow branches in a T-junction.
Contractions	Sudden	$0.5(1 - \beta^2)$
	Gradual, angle of convergence $\alpha < 22^\circ$	$0.8\sin(\alpha)(1 - \beta)$
	Gradual, angle of convergence $\alpha > 22^\circ$	$0.5\sqrt{\sin(\alpha)}(1 - \beta)$
Expansions	Sudden	$(1 - \beta^2)^2$
	Gradual, angle of convergence $\alpha < 22^\circ$	$2.6\sin(\alpha)(1 - \beta)^2$
	Gradual, angle of convergence $\alpha > 22^\circ$	$(1 - \beta)^2$

Above, β is the ratio of small to large cross-sectional area. The point friction losses listed above apply for Newtonian fluids. Point losses applying to non-Newtonian flow can be added as user-defined expressions, for instance from (Ref. 16).

For the point pressure loss features Bends, Valves, T-junctions, Y-junctions, n-way junctions, Contractions, and Expansions, the pressure loss can also be specified as a pressure drop Δp directly.

T-junctions

Several options to specify pressure drop across the T-junction branches are available. If the **Loss coefficients** option is selected, the energy loss between main branches and junction and the energy loss between the side branch and junction are calculated as

$$\left| p_{\text{main}, 1} + \frac{1}{2}\rho_{\text{main}, 1}u_{\text{main}, 1}^2 - p_{\text{junction}} \right| = \frac{1}{2}K_{\text{main}}\rho_{\text{main}, 1}u_{\text{main}, 1}^2$$

$$\left| p_{\text{main}, 2} + \frac{1}{2}\rho_{\text{main}, 2}u_{\text{main}, 2}^2 - p_{\text{junction}} \right| = \frac{1}{2}K_{\text{main}}\rho_{\text{main}, 2}u_{\text{main}, 2}^2$$

$$\left| p_{\text{side}} + \frac{1}{2}\rho_{\text{side}}u_{\text{side}}^2 - p_{\text{junction}} \right| = \frac{1}{2}K_{\text{side}}\rho_{\text{side}}u_{\text{side}}^2 .$$

Note that for the Nonisothermal Pipe Flow and Reacting Pipe Flow interfaces, the respective pressure drops are calculated as

$$\left| p_{\text{main}, 1} - p_{\text{junction}} \right| = \frac{1}{2}K_{\text{main}}\rho u_{\text{main}, 1}^2$$

$$\left| p_{\text{main}, 2} - p_{\text{junction}} \right| = \frac{1}{2}K_{\text{main}}\rho u_{\text{main}, 2}^2$$

$$\left| p_{\text{side}} - p_{\text{junction}} \right| = \frac{1}{2}K_{\text{side}}\rho u_{\text{side}}^2 .$$

The **Loss coefficient with respect to common branch** option, which is available for the Pipe Flow interface, implements the loss coefficients according to Ref. 28. The pressure loss for a T-junction is often expressed in terms of the flow in the common branch. For joining flows, the common branch is the collector branch (Figure 2-2, left)

$$p_b + \frac{1}{2}\rho_b u_b^2 = p_{\text{common}} + \frac{1}{2}\rho_{\text{common}}u_{\text{common}}^2 + \frac{1}{2}K_{b, \text{common}}^j \rho_{\text{common}}u_{\text{common}}^2$$

where $K_{b,\text{common}}^j$ is the loss coefficient between the branch b and the common branch for joining flows.

For separating flow, the common branch is the supplier branch

$$p_b + \frac{1}{2}\rho_b u_b^2 = p_{\text{common}} + \frac{1}{2}\rho_{\text{common}} u_{\text{common}}^2 - \frac{1}{2}K_{b,\text{common}}^s \rho_{\text{common}} u_{\text{common}}^2$$

For converging (joining) flow, where the collector branch is the side branch (Figure 2-2, left)

$$K_{ms}^j = 0.81 - 0.95q_{bc} + 1.34q_{bc}^2$$

For converging flow, where the collector branch is the main branch (Figure 2-2, right)

$$K_{mm}^j = 1 - 0.95q_{bc} - 0.16(q_{bc} - q_{bc}^2) - 0.46(1 - q_{bc})$$

$$K_{sm}^j = -1 + 3.38q_{bc} + ((1 - 0.5d_{bc}^{1.3})d_{bc}^4 - 1.84q_{bc}^2)q_{bc}^2$$

For diverging (separating) flow, where the supplier branch is the side branch (Figure 2-3, left)

$$K_{sm}^s = 1 - 1.18q_{bc} - 0.68q_{bc}^2$$

For diverging flow, where the supplier branch is the main branch (Figure 2-3, right)

$$K_{mm}^s = 0.36 - 0.98q_{bc} + 0.62q_{bc}^2 + 0.03q_{bc}^8$$

$$K_{ms}^s = 1 - 1.13q_{bc} + (0.81 + (1.12d_{bc} - 1.08d_{bc}^3 + 0.57)d_{bc}^4)q_{bc}^2$$

The mass flow rate ratio $q_{bc} = q_b/q_{\text{common}}$ and the hydraulic diameters ratio $d_{bc} = d_{hb}/d_{h\text{common}}$ are computed automatically. Only junction with sharp corners are considered.

The **Loss coefficients, extended model** option, that is available for the Pipe Flow interface, allows you to specify the loss coefficient in more details and account for the

flow directions. Enter values or expressions for the six dimensionless loss coefficients. See [Figure 2-2](#)-[Figure 2-3](#).

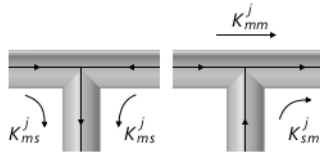


Figure 2-7: T-junction, converging flow.

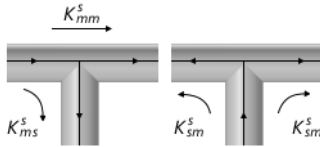


Figure 2-8: T-junction, diverging flow.

The losses for the outgoing branches are calculated as

$$p_{av,in}^0 - p_{j,out} - \frac{1}{2} \rho_{j,out} u_{j,out}^2 = \sum_{i,in} \frac{1}{2} K_{i,in,j,out} \rho_{i,in} u_{i,in}^2$$

The total pressure is assume to be equal for all incoming branches:

$$p_{av,in}^0 = p_{j,out} + \frac{1}{2} \rho_{j,out} u_{j,out}^2$$

Use the **Pressure drops** option to specify a value or expression for the pressure drop explicitly for each branch respectively:

$$\Delta p_{1,s} = p_{main,1} - p_{side}$$

$$\Delta p_{2,s} = p_{main,2} - p_{side}$$

Main 1 is the main pipe segment that has the lower index of the two in the selection list (for example, see the list in Pipe Properties).

Orifice

A restrictive orifice is a fixed-geometry element installed in a pipeline that creates a pressure drop for control, protection, or balancing purposes. The pressure drop due to presence of the orifice in the pipe network is calculated as:

$$\Delta p = \frac{1}{2} K \rho u_{in}^2$$

where the loss coefficient based on the inlet pipe velocity K related to the loss coefficient relative to the orifice K_o as

$$K = K_o \left(\frac{A_{in}}{A_o} \right)^2$$

Predefined loss coefficients are available for both sharp-edged and rounded-edge orifices. (Figure 2-9-Figure 2-10).

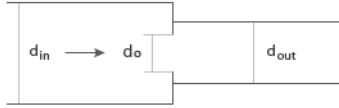


Figure 2-9: Sharp edged orifice.

The following expression is used for sharp orifices (Ref. 28)

$$K_o = 0.0696(1 - \beta^5)\lambda^2 + \left[\lambda - \left(\frac{d_o}{d_{out}} \right)^2 \right]^2$$

where β is diameter ratio $\beta = d_o/d_{in}$ and the jet contraction coefficient λ is defined as

$$\lambda = 1 + 0.622(1 - 0.215\beta^2 - 0.785\beta^5)$$

For gently rounded orifice with $r_o \geq d_o$, the loss coefficient can be found from (Ref. 28)

$$K_o = 0.030(1 - \beta)(1 - \beta^5) + \left[\lambda - \left(\frac{d_o}{d_{out}} \right)^2 \right]^2$$

Otherwise, the coefficient equals

$$K_o = 0.0696 \left(1 - 0.569 \frac{r_o}{d_o} \right) \left(1 - \sqrt{\frac{r_o}{d_o}} \beta \right) (1 - \beta^5) \lambda^2 + \left[\lambda - \left(\frac{d_o}{d_{out}} \right)^2 \right]^2$$

where the contraction coefficient is given by

$$\lambda = 1 + 0.622 \left(1 - 0.30 \sqrt{\frac{r_o}{d_o}} - 0.7085 \frac{r_o}{d_o} \right) (1 - 0.215\beta^2 - 0.785\beta^5)$$

Note that, the diameter ratio is constrained by geometry: the achievable rounding radius is limited by the radial clearance between the pipe wall and the orifice.

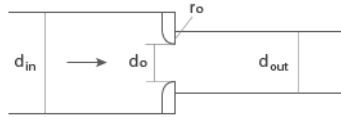


Figure 2-10: Round edged orifice.

Control Valve

The pressure drop across a control valve can be characterized using flow coefficients such as C_v , K_v , or the resistance coefficient, K . The C_v and K_v coefficients relate the flow rate to the pressure drop under standardized conditions in imperial and metric units, respectively, while the dimensionless coefficient K expresses pressure loss as a function of velocity head. These formulations offer equivalent approaches for assessing valve characteristics. Valve opening refers to the adjustment of the closure element (for example, plug, disc, or ball) relative to the flow passage, which changes the effective flow area. As the valve opens, resistance to flow decreases and capacity increases. Control valve inherent characteristics define the relationship between valve opening and flow capacity under constant pressure drop conditions.

The flow of liquid through a valve can be described by the following industry-standard equation:

$$Q_v = C_v f_{ch}(x_{fr}) \sqrt{\Delta p \frac{\rho_{sc}}{\rho}}$$

Here, Q_v is the volumetric flow rate in (gallons per minute in the U.S. customary system); C_v is the flow coefficient, the flow in gallons per minute that flows through a valve that has a pressure drop of 1 psi across the valve; Δp is the pressure drop across the valve in psi; ρ_{sc} is the density of water at 60°F; x_{fr} is the fraction of valve opening; and f_{sc} is the inherent flow characteristic.

To calculate the pressure drop across the valve, the dimensionless loss coefficient K is used:

$$\Delta p = \frac{1}{2} K \rho \left(\frac{Q_v}{A} \right)^2$$

The relation between the flow coefficient C_v and the dimensionless loss coefficient is given by

$$K = 1444.6 \left(\frac{A}{A_r} \right)^2 \frac{1}{\tilde{C}_v^2}$$

where \tilde{C}_v is the numerical value (dimensionless) of the flow coefficient C_v and $A_r = 1 \text{ in}^2$.

Another commonly used coefficient is the flow factor K_v , which represents the volume of water (in m^3/h) flowing through a valve at 16°C with a pressure drop of 1 bar:

$$K = 1081.015 \left(\frac{A}{A_r} \right)^2 \frac{1}{\tilde{K}_v^2}$$

where \tilde{K}_v is the numerical value (dimensionless) of the flow factor K_v .

The characteristic is determined by the valve's internal geometry and trim design. Common inherent characteristics include linear, equal-percentage, and quick-opening.

For a linear characteristic, the flow rate increases directly with valve lift

$$f_{\text{ch}} = x_{\text{fr}}$$

For an equal percentage valve, each increment of valve opening produces the same percentage change in flow:

$$f_{\text{ch}} = R^{x_{\text{fr}}}$$

Here, R is the rangeability factor, a valve design parameter that is usually in the range [20, 50].

For a valve with quick opening characteristics

$$f_{\text{ch}} = x_{\text{fr}}^n, \quad n > 1$$

For linear or quick-opening flow characteristics, the variation in loss coefficients is expressed as

$$K = \frac{K_{\min}}{\left[f_{\text{ch}} + \sqrt{\frac{K_{\min}}{K_{\max}}} (1 - f_{\text{ch}}) \right]^2}$$

For equal-percentage valves, the following expression is used

$$K = K_{\min} R^{2(1 - x_{\text{fr}}^n)}$$

A volumetric flow rate can be specified

$$Q_v = Q_{\text{vvalve}}$$

In this case the corresponding pressure drop is calculated to satisfy the flow condition. Using the specified flow rate and the resulting pressure loss, the values of K , C_v , and K_v are computed and available as postprocessing variables. The pressure loss must always be positive, as a valve cannot operate as a pump. However, under certain conditions, the control valve feature may introduce a pressure gain. If this occurs, the values of K , C_v , and K_v are set to -1 .

3-Way Valve

A three-way control valve regulates fluid mixing or diversion between two streams. Its inherent characteristics, determined by port geometry and trim design, define how flow capacity varies with valve position under constant pressure drop conditions (Figure 2-11). The flow is always present in the main branch. When the opening fraction $x_{\text{fr}} = 1$, the flow is fully directed from the main branch to side branch 1. For $0 < x_{\text{fr}} < 1$, the flow is split between side branch 1 and side branch 2. When $x_{\text{fr}} = 0$, the flow is fully directed from the main branch to side branch 2.

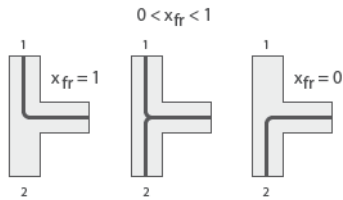


Figure 2-11: 3-way valve.

The side ports of a 3-way valve are linked such that as one port opens, the other closes

$$\begin{aligned}x_{fr1} &= x_{fr} \\x_{fr2} &= 1 - x_{fr}\end{aligned}$$

In a 3-way valve, each port can be assigned a different flow characteristic. Pressure loss for each side is calculated in the same way as for a control valve; see [Control Valve](#) for details.

References for the Pipe Flow Interface

1. S.W. Churchill, “Friction factor equation spans all fluid-flow regimes,” *Chem. Eng.*, vol. 84, no. 24, pp. 91–92, 1997.
2. D.J. Wood, “An explicit friction factor relationship,” *Civil Engrs.*, ASCE 60, 1966.
3. S.E. Haaland, “Simple and explicit formulas for the friction factor in turbulent flow,” *J. Fluids Engineering (ASME)*, vol. 103, no. 5, pp. 89–90, 1983.
4. C.F. Colebrook, “Turbulent flow in pipes with particular reference to the transition region between the smooth and rough pipe laws,” *J. Inst. Civil Engineers*, London, vol. 11, pp. 133–156, 1939.
5. R.J. Houghtalen, N.H.C. Hwang, and A.O. Akan, *Fundamentals of Hydraulic Engineering Systems*, 4th ed., Prentice Hall, p. 63, 2009.
6. P.K. Swamee and A.K. Jain, “Explicit Equations for Pipe-flow Problems,” *J. Hydraulics Division (ASCE)*, vol. 102, no. 5, pp. 657–664, 1976.
7. A.B. Metzner and J.C. Reed, “Flow of non-Newtonian fluids — correlation of the laminar, transition, and turbulent-flow regions,” *AIChE Journal*, vol. 1, no. 4, pp. 434–440, 1955.
8. T.F. Irvine, *Chem Eng Commun*, vol. 65, p. 39, 1988.
9. N.W. Ryan and M.M. Johnson, “Transition from laminar to turbulent flow in pipes,” *AIChE Journal*, vol. 5, no. 4, pp. 433–435, 1959.
10. P.K. Swamee and N. Aggarwal, “Explicit equations for laminar flow of Bingham plastic fluids,” *J. Petroleum Science and Engineering*, vol. 76, no. 3–4, pp. 178–184, 2011.
11. R. Darby, R. Mun, and D.V. Boger, “Predict Friction Loss in Slurry Pipes,” *Chem. Eng.*, vol. 99, p. 116, 1992.

12. P.K. Swamee and N. Aggarwal, "Explicit equations for laminar flow of Herschel-Bulkley fluids," *Canadian Journal of Chemical Engineering*, vol. 89, no. 6, pp. 1426–1433, 2011.
13. R.P. Chhabra and J.F. Richardson, *Non-Newtonian Flow in the Process Industries: Fundamentals and Engineering Applications*, Butterworth Heinemann, p. 111, 2004.
14. J.M. Coulson and J.F. Richardson, *Chemical Engineering, Volume. 1*, 4th ed., Pergamon Press, p. 56, 1990.
15. *Engineering and Design Liquid Process Piping*, Department of the Army, U.S. Army Corps of Engineers, Engineering Manual No.1110-1-4008, 1999.
16. R.P. Chhabra and J.F. Richardson, *Non-Newtonian Flow in the Process Industries: Fundamentals and Engineering Applications*, Butterworth Heinemann, p. 111 and 140–149, 2004.
17. C.L. Barnard and others, "A Theory of Fluid Flow in Compliant Tubes," *Biophysical Journal*, vol. 6, no. 6, pp. 717–724, 1966.
18. F.P. Incropera and D.P. DeWitt, *Fundamentals of Heat and Mass Transfer*, 4th ed., John Wiley & Sons, 1996.
19. V. Gnielinski, "New equations for heat and mass transfer in turbulent pipe and channel flow," *Int. Chem. Eng.*, vol. 16, pp. 359–368, 1976.
20. S.W. Churchill and M. Bernstein, "A Correlating Equation for Forced Convection From Gases and Liquids to a Circular Cylinder in Crossflow," *J. Heat Transfer*, vol. 99, no. 2, pp. 300–307, 1977.
21. S.W. Churchill and H.H.S. Chu, "Correlating Equations for Laminar and Turbulent Free Convection from a Vertical Plate," *Int. J. Heat Mass Transfer*, vol. 18, no. 11, pp. 1323–1329, 1975.
22. G. Taylor, "Dispersion of Soluble Matter in Solvent Flowing Slowly through a Tube," *Proc. Roy. Soc. A.*, vol. 219, pp. 186–203, 1953.
23. G. Taylor, "The Dispersion of Matter in Turbulent Flow Through a Pipe," *Proc. Roy. Soc. A.*, vol. 223, pp. 446–468, 1954.
24. L.T. Fan and C.B. Wang, "Dispersion of Mater in Non-Newtonian Laminar Flow Through a Circular Tube," *Proc. Roy. Soc. A.*, vol. 292, pp. 203–208, 1966.

25. M.V. Lurie, *Modeling of Oil Product and Gas Pipeline Transportation*, WILEY-VCH Verlag GmbH & Co., KGaA, Weinheim, 2008.
26. R. Balasubramaniam and others, *Two Phase Flow Modeling: Summary of Flow Regimes and Pressure Drop Correlations*, NASA Report No. NASA/CR—2006-214085, National Center for Space Exploration Research, Cleveland, OH.
27. M. Ungarish, *Hydrodynamics of Suspensions*, Springer-Verlag, Berlin, p. 18, ISBN 3540547622 (Berlin), 0387547622 (New York), 1993.
28. D.C. Rennels and H.M. Hudson, *Pipe flow, A Practical and Comprehensive Guide*, John Wiley & Sons, 2012.

Theory for the Water Hammer Interface

In this section:

- [Flow Equations](#)
- [Reference for the Water Hammer Interface](#)

Flow Equations

The Water Hammer interface implements both the continuity equation and the momentum equations for a compressible fluid traveling inside pipes of variable cross section.

THE CONTINUITY EQUATION

The mass conservation for a fluid inside a pipe is given by:

$$\frac{\partial A\rho}{\partial t} + \nabla \cdot (A\rho\mathbf{u}) = 0 \quad (2-45)$$

where A (SI unit: m^2) is the cross-section area of the pipe, ρ (SI unit: kg/m^3) is the fluid density, and \mathbf{u} (SI unit: m/s) is the tangential fluid velocity.

For adiabatic processes, the density and cross-section area are functions of the pressure, so the continuity equation reads

$$\frac{\partial A(p)\rho(p)}{\partial t} + \nabla \cdot (A(p)\rho(p)\mathbf{u}) = 0 \quad (2-46)$$

In a first-order approximation, this equals

$$A_0\rho_0\left(\frac{1}{K_p} + \frac{1}{K_A}\right)\frac{\partial p}{\partial t} + \nabla \cdot (A_0\rho_0\mathbf{u}) = 0 \quad (2-47)$$

where K_p is the bulk modulus of the fluid (the inverse of its compressibility), and K_A is the effective bulk modulus of the cross section area. A_0 and ρ_0 are the reference area and reference density at a given pressure p_0 .

The Water Hammer wave speed c (SI unit: m/s) is given by a combination of fluid and structural material properties

$$\frac{1}{c^2} = \rho_0 \left(\frac{1}{K_p} + \frac{1}{K_A} \right) \quad (2-48)$$

The effective bulk modulus for the cross-sectional area K_A (SI unit: Pa) is given by the pipe's material properties

$$K_A = E \frac{w_{th}}{d_h}$$

where E is the Young's modulus, d_h is the hydraulic diameter, and w_{th} is the pipe's wall thickness. This is the so-called Korteweg formula ([Ref. 1](#))

The Korteweg formula can also be extended to pipes suffering from axial stresses. In this case, a more general formula would include Poisson's ratio, ν , of the pipe's material

$$K_A = \frac{E w_{th}}{\zeta d_h} \quad (2-49)$$

where $\zeta = 1$ for pipe with zero axial stress (this is Korteweg's original formula for a pipe furnished with expansion joints), $\zeta = 1 - \nu/2$ for a pipe anchored at one end, and $\zeta = 1 - \nu^2$ for a pipe anchored at both ends.

THE MOMENTUM EQUATION

The momentum equation is written as:

$$\rho \frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_D \frac{\rho}{2d_h} \mathbf{u} |\mathbf{u}| + \mathbf{F} \quad (2-50)$$

here, f_D is the Darcy friction factor, normally a function of the Reynolds number, the surface roughness and the hydraulic diameter, as described in [Expressions for the Darcy Friction Factor](#).

This set of equations are normally acknowledged as the Water Hammer equations, and are they mainly written in the literature as (consider gravity forces, so $\mathbf{F} = \rho \mathbf{g}$)

$$A \frac{1}{c^2} \frac{\partial p}{\partial t} + \nabla \cdot (A \rho \mathbf{u}) = 0 \quad (2-51)$$

$$\rho \frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_D \frac{\rho}{2d_h} \mathbf{u} |\mathbf{u}| + \rho \mathbf{g} \quad (2-52)$$


BOUNDARY CONDITIONS

The available boundary conditions are Closed, Pressure, Velocity, and [Additional Flow Resistances](#).

Reference for the Water Hammer Interface

I. M. Ghidaoui and others, “A Review of Water Hammer Theory and Practice,” *Appl. Mech. Rev.*, vol. 58, pp. 49–76, 2005.

Heat Transfer Interfaces



This physics interface, which is found under the **Heat Transfer** branch () when adding a physics interface, has functionality for simulating heat transfer in pipe networks, including wall heat transfer to the surroundings.

In this chapter:

- [The Mechanisms for Heat Transfer](#)
- [The Heat Transfer in Pipes Interface](#)
- [Theory for the Heat Transfer in Pipes Interface](#)

See [Heat Transfer](#) in the *COMSOL Multiphysics Reference Manual* for other Heat Transfer interface and feature node settings.

The Mechanisms for Heat Transfer

The **Heat Transfer in Pipes** interface () , found under the **Heat Transfer** branch () when adding a physics interface, solves a temperature equation for a fluid transported in a pipe. The physics interface requires the fluid velocity as input, provided either by the user or by coupling to the Pipe Flow interface. However, it is advisable to instead use [The Nonisothermal Pipe Flow Interface](#) which is a multiphysics interface that combines Pipe Flow and Heat Transfer in Pipes to model the velocity, pressure, and temperature fully coupled.





The **Heat Transfer in Solids**, **Heat Transfer in Fluids** (general convection and conduction, nonisothermal flow, and conjugate heat transfer), and **Joule Heating** interfaces all belong to the COMSOL Multiphysics base package. See [Heat Transfer](#) in the *COMSOL Multiphysics Reference Manual* for more information.

COUPLING TO OTHER PHYSICS INTERFACES

It is often relevant to couple heat transport in pipe networks to heat transfer in the region surrounding the pipe network. Simulating the cooling of a mold is a good example of such an application. The Heat Transfer in Pipes interface or the Nonisothermal Pipe Flow interface can be used to model the removal of heat in the cooling channels represented by 1D edges, and couple that heat loss to a 3D model of the mold. The pipe heat model is automatically coupled to the 3D surroundings.

More extensive descriptions of heat transfer, such as 3D turbulent flow models or problems involving surface-to-surface radiation, can be found in the Heat Transfer Module. Furthermore, some cases that involve geothermal applications with groundwater flow models for porous media are better handled by the Subsurface Flow Module.

The Heat Transfer in Pipes Interface

The **Heat Transfer in Pipes (htp)** interface () , found under the **Heat Transfer** branch () when adding a physics interface, is used to model heat transfer by conduction and convection in pipes and channels of different shapes, where the fluid velocity and pressure fields are known a priori. The interface uses 1D energy balance to determine the temperature profiles on curve segments, or lines. These lines can be drawn in 2D or 3D and represent simplifications of hollow tubes. Wall heat transfer, including multilayer walls and cladding is included as an option.

The temperature equation corresponds to a 1D convection–diffusion equation that may contain additional contributions like heat sources.

The [Nonisothermal Pipe Flow Interface](#) extends this physics interface by providing equations to compute the velocity and pressure fields when they are unknown.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Heat Transfer, Pipe Properties, Temperature, and Initial Values**. Then, from the **Physics** toolbar, add other nodes that implement, for example, boundary conditions. You can also right-click **Heat Transfer in Pipes** 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 `htp`.

DEPENDENT VARIABLES

This physics interface defines the **Temperature** T (SI unit: K) dependent variable (field). If required, edit the name, but dependent variables must be unique within a model.

DISCRETIZATION

The default discretization for temperature is quadratic.



- [Edge, Boundary, Point, and Pair Nodes for the Heat Transfer in Pipes Interface](#)
- [Theory for the Heat Transfer in Pipes Interface](#)
- [Handling Frames in Heat Transfer in the *COMSOL Multiphysics Reference Manual*](#)

Edge, Boundary, Point, and Pair Nodes for the Heat Transfer in Pipes Interface

The [Heat Transfer in Pipes Interface](#) has these 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.

- [Heat Outflow](#)
- [Heat Source](#)
- [Heat Transfer](#)
- [Initial Values](#)
- [Pipe Properties](#)
- [Temperature](#)
- [Wall Heat Transfer](#)
- [Open Pipe End](#)



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.

Heat Transfer

Use the **Heat Transfer** node to define the tangential velocity, density, dynamic viscosity, and heat convection and conduction properties.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) uses the value **From material**. For **User defined** enter a different value or expression.

The default **Dynamic viscosity** μ (SI unit: Pa·s) uses the value **From material** and describes the relationship between the shear rate and the shear stresses in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick (such as oil) have a higher viscosity.

HEAT CONVECTION AND CONDUCTION

Select an option from the **Tangential velocity** list — **User defined**, **Tangential velocity (nipfl/nipfl)**, or **Tangential velocity (pfl/pfl)**. For **User defined** enter a value for the **Tangential velocity** u (SI unit: m/s).

The default **Density** ρ (SI unit: kg/m^3), **Dynamic viscosity** μ (SI unit: Pa·s), **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)), **Ratio of specific heats** γ (dimensionless), and **Thermal conductivity** k (SI unit: W/(m·K)) all use values **From material**. For **User defined** enter different values or expressions.

Heat Source

The **Heat Source** describes heat generation within the domain. Express heating and cooling with positive and negative values, respectively. Add one or more nodes as needed — all heat sources within a domain contribute to the total heat source.

HEAT SOURCE

Click the **General source**, **Linear source**, or **Heat rate** button.

- For **General source** enter a value for the distributed heat source Q (SI unit: W/m^3).
- For **Heat rate** enter a value for the heat rate P_0 (SI unit: W). In this case $Q = P_0/L$ where L is the total length of the selected boundaries.
- For **Linear source** ($Q = q_s \cdot T$) enter the **Linear heat source** q_s (SI unit: $\text{W}/(\text{m}^3 \cdot \text{K})$).



The advantage of writing the source in this form is that it can be stabilized by the streamline diffusion. The theory covers q_s that is independent of the temperature, but some stability can be gained as long as q_s is only weakly dependent on the temperature.



Wall Heat Transfer

Use the **Wall Heat Transfer** node to set up heat exchange across the pipe wall. Define the external temperature and the nature of the heat transfer. The **Internal Film Resistance**, **Wall Layer**, and **External Film Resistance** subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar, **Attributes** menu. These compute heat transfer coefficients based on the section [Theory for the Heat Transfer in Pipes Interface](#).



You must add at least an Internal Film Resistance subnode. This corresponds to a drilled channel in a solid. If a pipe wall exists, (pipe embedded in a solid), add also Wall Layer. If the pipe wall has more than one layer, add more Wall Layer features. If the pipe exchanges heat to a surrounding fluid by forced or natural convection, add also External Film Resistance.

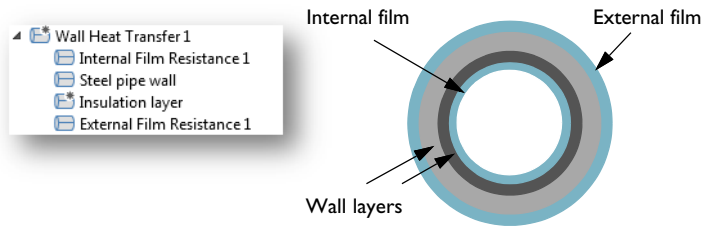


Figure 3-1: Internal and external films and the associated film coefficients (blue), and wall layers (dark and light gray).

HEAT TRANSFER MODEL

Enter a value for the **External Temperature** T_{ext} (SI unit: K). If an expression is used for T_{ext} that is a locally defined variable on the edge, the checkbox **Take average in adjacent domain** should be cleared.

Use the [Pipe Wall Heat Transfer](#) multiphysics coupling if the temperature in surrounding 3D solid physics is calculated using **Heat Transfer** interface.

Internal Film Resistance

The **Internal Film Resistance** subnode is available from the context menu (right-click the [Wall Heat Transfer](#) parent node) or from the **Physics** toolbar's **Attributes** menu. Use it to define the film resistance.



Even if you do not expect a boundary layer to build up, the thermal film theory works well for laminar flow profiles. In fact, there is an analytical solution available to the Nusselt number for laminar flow in circular tubes ($Nu = 3.66$).

FILM RESISTANCE

The default **Internal film resistance model** is **Automatic** and the Nusselt number is computed as described in the section [Theory for the Heat Transfer in Pipes Interface](#). For **User defined** enter a value for the **Nusselt number** (dimensionless). The default is 3.66.

External Film Resistance

The **External Film Resistance** subnode is available from the context menu (right-click the [Wall Heat Transfer](#) parent node) or from the **Physics** toolbar's **Attributes** menu. Use it to define the external film resistance features including the material, thermal conductivity, density, external velocity, and pressure of the external fluid. You can select between forced and natural convection.

SPECIFICATION

Select an **External film heat transfer model** — **External forced convection** (the default), **External natural convection**, or **User defined**.

External Forced Convection and External Natural Convection

For **External forced convection** or **External natural convection** the **Surrounding fluid** uses the **Domain material** by default.



Be careful to select the right material in the **Settings** window for **External Film Resistance**. Many times this should be another material than the **Domain material** which is the internal fluid.

The default **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)), **Thermal conductivity** k (SI unit: W/(m·K)), **Density** ρ (SI unit: kg/m³), and **Dynamic viscosity** μ (SI unit: Pa·s), all use values **From material**. For **User defined** enter different values or expressions than the default, which is 0 for all options.

The **External velocity** u_{ext} (SI unit: m/s) is **User defined** by default. This is the velocity of the cooling or heating fluid outside the pipe.

For **External natural convection** only, also enter a value or expression for the **External pressure** p_{ext} (SI unit: Pa). The default is 1 atm.



The natural convection correlations (see [External Film Resistance](#) under [The Heat Transfer Equation](#) in the section [Theory for the Heat Transfer in Pipes Interface](#)) require a temperature dependent density, or else the coefficient of thermal expansion β evaluates to 0 ([Equation 3-28](#)), which might generate unexpected results or even unstable models. Also, make sure that the density correlation $\rho(T)$ is smooth throughout the temperature interval used, or else β is discontinuous and can cause numerical instability.

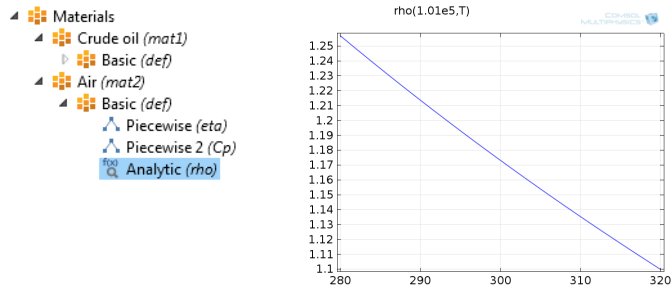


Figure 3-2: Before you use the Natural Convection option, it is good practice to select Density in the used material, and click Plot in the Settings window. That way you can inspect the behavior and make sure it is smooth within the temperature interval you plan to use it.

User Defined

For **User defined**:

- Enter the **Nusselt number** (dimensionless). The default is 3.66.
- The **Surrounding fluid** uses the **Domain material** by default.

- The **Thermal conductivity** k (SI unit: W/(m·K)) takes its value **From material**. For **User defined** enter a different value or expression. The default is 0 W/(m·K).
- Enter a value or expression for the **External pressure** p_{ext} (SI unit: Pa). The default is 1 atm).

Wall Layer

The **Wall Layer** subnode is available from the context menu (right-click the **Wall Heat Transfer** parent node) or from the **Physics** toolbar's **Attributes** menu. Use it to define the thermal conductivity and wall thickness. You can add more layers.

SPECIFICATION

Select an option from the **Thermal conductivity** list k (SI unit: W/(m·K)) — **Not set** (the default) or **User defined** to enter a different value or expression. The default is 0 W/(m·K).

Select an option from the **Wall thickness** list Δw (SI unit: m) — **Not set** (the default) or **User defined**. For **User defined** enter a value or expression. The default is 0 m.



You must add at least an Internal Film Resistance subnode. This corresponds to a drilled channel in a solid. If a pipe wall exists, (pipe embedded in a solid), add also Wall Layer. If the pipe wall has more than one layer, add more Wall Layer features.

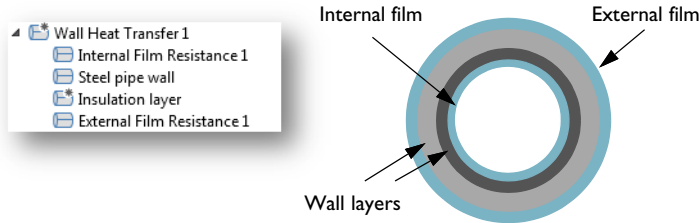


Figure 3-3: Internal and external films and the associated film coefficients (blue), and wall layers (dark and light gray).

Wall Heat Transfer, Inner

Use the **Wall Heat Transfer, Inner** node to set up heat exchange across the pipe wall. This feature is only applicable if **Annular** is selected as **Pipe Shape** in the **Pipe Properties** node. Define the core temperature T_{core} and the nature of the heat transfer. The [Internal Film Resistance](#) and [Wall Layer](#) subnodes are available from the context menu (right-click the parent node) or from the **Physics** toolbar's **Attributes** menu. These compute heat transfer coefficients based on the section [Theory for the Heat Transfer in Pipes Interface](#).



You must add at least an Internal Film Resistance subnode. This corresponds to a drilled channel in a solid. If a pipe wall exists, (pipe embedded in a solid), add also Wall Layer. If the pipe wall has more than one layer, add more Wall Layer features.

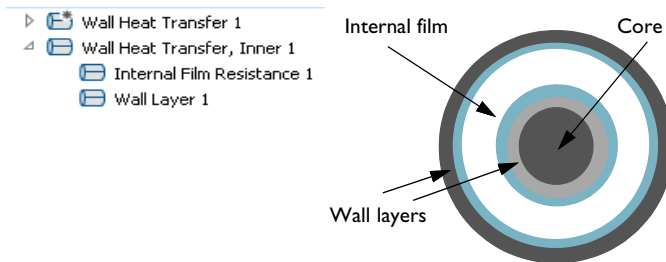


Figure 3-4: Internal film (blue) and wall layer (light gray).

Initial Values

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

INITIAL VALUES

Enter values or expressions for the initial value of the **Temperature** T (SI unit: K).


Temperature

Use the **Temperature** node to specify the temperature at an inlet of the pipe system.

TEMPERATURE

The equation for this condition is $T = T_0$, where T_0 is the prescribed temperature (SI unit: K) on the boundary. Enter the value or expression for the **Temperature** T_{in} . The default is 293.15 K.

CONSTRAINT SETTINGS

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

Heat Outflow

The **Heat Outflow** node provides a suitable boundary condition for convection-dominated heat transfer at outlet boundaries. In a model with convective heat transfer, this condition states that the only heat transfer over a boundary is by convection. The temperature gradient in the normal direction is zero, and there is no radiation. This is usually a good approximation of the conditions at an outlet boundary in a heat transfer model with fluid flow.

Open Pipe End

At the open pipe end both inflow and outflow conditions are supported. For an outgoing fluid flow across the boundary, it applies a zero conductive flux condition:

$$-\mathbf{e}_t \cdot \mathbf{q} = 0, \quad uI_e \geq 0 \quad (3-1)$$

For an incoming flow of velocity field u across the boundary, it accounts by default for the heat flux induced by the flow rate through a Danckwerts condition on the enthalpy:

$$-\mathbf{e}_t \cdot \mathbf{q} = A\rho\Delta H u I_e, \quad uI_e < 0 \quad (3-2)$$

where ΔH is the sensible enthalpy. See [Theory for the Open Pipe End Condition](#) for more details on the Danckwerts condition and the definition of ΔH .

UPSTREAM PROPERTIES

The conditions is based on the knowledge of the upstream temperature at the outer boundary of the virtual domain. Enter the Upstream temperature T_{ustr} outside the open boundary.

Theory for the Heat Transfer in Pipes Interface

The [Heat Transfer in Pipes Interface](#) and [The Reacting Pipe Flow Interface](#) theory is described in this section:

- [The Heat Transfer Equation](#)
- [Reference for the Heat Transfer in Pipes Interface](#)

The Heat Transfer Equation

This physics interface solves an energy balance equation for 1D pipes, taking the flow velocity as input.

HEAT BALANCE EQUATION

The energy equation for an incompressible fluid flowing in a pipe is ([Ref. 25](#)):

$$\rho A C_p \frac{\partial T}{\partial t} + \rho A C_p \mathbf{u} \cdot \nabla T = \nabla \cdot A k \nabla T + f_D \frac{\rho A}{2d_h} |\mathbf{u}|^3 + Q + Q_{\text{wall}} + Q_p \quad (3-3)$$

where ρ is the fluid density (SI unit: kg/m^3), A is the pipe cross section area (SI unit: m^2) available for flow, C_p (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$) is the heat capacity at constant pressure, T (SI unit: K) is the temperature. \mathbf{u} is a velocity field. For information about the tangential velocity in pipe flow, see [Theory for the Pipe Flow Interface](#). Further, k (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) is the thermal conductivity. The second term on the right hand side corresponds to friction heat dissipated due to viscous shear. Q (SI unit: W/m) represents a general heat source and Q_{wall} (SI unit: W/m) represents external heat exchange through the pipe wall. Note that the Q_{wall} term is detailed below.

An additional term Q_p can be added to the right-hand side of the equation by enabling the [Pressure Work](#) checkbox:

$$Q_p = -\frac{TA}{\rho} \left(\frac{\partial \rho}{\partial T} \right) \left(\frac{\partial p}{\partial T} + \mathbf{u} \cdot \nabla p \right) \quad (3-4)$$

This term is optional and can be used if the pressure drop is expected to be considerable and the fluid is compressible. The contribution follows the same theory

as the pressure work term described in the section [The Nonisothermal Flow and Conjugate Heat Transfer Equations](#) in the *Heat Transfer Module User's Guide*.

Wall Heat Transfer

The radial heat transfer from the surroundings into the pipe is given by

$$Q_{\text{wall}} = (hZ)_{\text{eff}}(T_{\text{ext}} - T) \text{ (W/m)} \quad (3-5)$$

In [Equation 3-5](#), $(hZ)_{\text{eff}}$ is an effective value of the heat transfer coefficient h (SI unit: $\text{W}/(\text{m}^2 \cdot \text{K})$) times the wall perimeter Z (SI unit: m) of the pipe. T_{ext} (SI unit: K) the external temperature outside of the pipe. See [Figure 3-5](#). Q_{wall} appears as a source term in the pipe heat transfer equation, [Equation 3-3](#).

The Wall Heat Transfer feature requires the external temperature and at least an internal film resistance subnode added to it. The individual contributions of heat transfer coefficients can be added by subnodes to the Wall Heat Transfer feature. The subnodes are:

- Internal Film Resistance
- Wall Layer
- External Film Resistance

T_{ext} in [Equation 3-5](#) can be a constant, parameter, or expression. Effective value of the heat transfer coefficient $(hZ)_{\text{eff}}$ is automatically calculated through film resistances and wall layers that are added as subnodes; see [Equation 3-18](#).

If external temperature is computed by 3D Heat Transfer interface, add a **Pipe Wall Heat Transfer** coupling. The temperature coupling between the pipe and the surrounding domain is implemented as a line heat source in the 3D domain. The source strength is proportional to the temperature difference ([Equation 3-5](#)) between the pipe fluid and the surrounding domain.

The overall heat transfer coefficient including internal film resistance, wall resistance and external film resistance can be deduced as follows, with reference to [Figure 3-5](#).

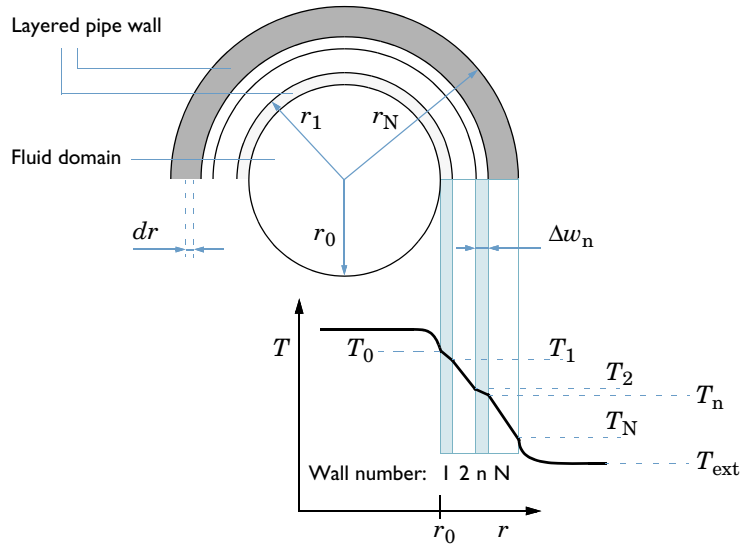


Figure 3-5: Temperature distribution across the pipe wall.

- r_n (SI unit: m) is the outer radius of wall n
- $w = r - r_0$ (SI unit: m) a wall coordinate, starting at the inner radius r_0
- $\Delta w_n = r_n - r_{n-1}$ (SI unit: m) the wall thickness of wall n
- Z_n (SI unit: m) is the outer perimeter of wall n
- h_{int} and h_{ext} are the film heat transfer coefficients on the inside and outside of the tube, respectively (SI unit: $W/(m^2 \cdot K)$).
- k_n is the thermal conductivity (SI unit: $W/(m \cdot K)$) of wall n

Shell balance In [Figure 3-5](#), consider a short length section ΔL of pipe, perpendicular to the figure plane. The heat leaving the internal fluid of that segment into the wall is

$$Q_0 = h_{int} A_Q \cdot (T - T_0) \text{ (W)} \quad (3-6)$$

Here, $A_Q = \Delta L 2\pi r_0$ (SI unit: m^2) is the area available for heat flux into the wall. For stationary conditions that same amount of heat must travel through any cylindrical shell at radius r in wall 1 (or any wall).

$$Q_1 = \Delta L 2\pi r \cdot \left(-k_1 \frac{dT}{dr}\right) \quad (3-7)$$

Rearrange and integrate from r_0 to r_1 .

$$\int_{T_0}^{T_1} dT = - \int_{r_0}^{r_1} \left(\frac{Q_1}{\Delta L 2\pi k_1} \cdot \frac{1}{r} \right) dr \quad (3-8)$$

Perform the integration

$$T_1 - T_0 = - \frac{Q_1}{\Delta L 2\pi k_1} \cdot \ln\left(\frac{r_1}{r_0}\right) \quad (3-9)$$

and rearrange

$$Q_1 = \frac{\Delta L 2\pi k_1}{\ln\left(\frac{r_1}{r_0}\right)} \cdot (T_0 - T_1). \quad (3-10)$$

For the example of two wall layers, the heat flow is equal across any shell from the inner bulk fluid to the surroundings, and we can set all $Q_n = Q$.

$$\begin{aligned} Q &= h_{\text{int}} \Delta L 2\pi r_0 \cdot (T - T_0) \\ Q &= \frac{\Delta L 2\pi k_1}{\ln(r_1/r_0)} \cdot (T_0 - T_1) \\ Q &= \frac{\Delta L 2\pi k_2}{\ln(r_2/r_1)} \cdot (T_1 - T_2) \\ Q &= h_{\text{ext}} \Delta L 2\pi r_2 \cdot (T_2 - T_{\text{ext}}) \end{aligned} \quad (3-11)$$

Substituting

$$\Theta = \Delta L 2\pi, \quad (3-12)$$

and making a linear combination of the equations [Equation 3-11](#) gives

$$Q = (hA_Q)_{\text{eff}} \cdot (T - T_{\text{ext}}) \quad (3-13)$$

where $(hA_Q)_{\text{eff}}$ is an effective conductance:

$$(hA_Q)_{\text{eff}} = \frac{1}{\frac{1}{r_0 h_{\text{int}} \Theta} + \frac{\ln(r_1/r_0)}{k_1 \Theta} + \frac{\ln(r_2/r_1)}{k_2 \Theta} + \frac{1}{r_2 h_{\text{ext}} \Theta}} \quad (3-14)$$

For the general case with N wall layers this reads

$$(hA_Q)_{\text{eff}} = \frac{1}{\frac{1}{r_0 h_{\text{int}} \Theta} + \sum_{n=1}^N \left(\frac{\ln\left(\frac{r_n}{r_{n-1}}\right)}{k_n \Theta} \right) + \frac{1}{r_N h_{\text{ext}} \Theta}} \quad (3-15)$$

Now let

$$(hZ)_{\text{eff}} = \frac{(hA_Q)_{\text{eff}}}{\Delta L}, \quad (3-16)$$

where Z (SI unit: m) is an average perimeter (circumference), of the pipe, taken over the thickness of the pipe walls. Combine [Equation 3-12](#) and [Equation 3-16](#) such that

$$(hA_Q)_{\text{eff}} = (hZ)_{\text{eff}} \Theta / (2\pi)$$

and insert in [Equation 3-15](#):

$$(hZ)_{\text{eff}} = \frac{2\pi}{\frac{1}{r_0 h_{\text{int}}} + \frac{1}{r_N h_{\text{ext}}} + \sum_{n=1}^N \left(\frac{\ln\left(\frac{r_n}{r_{n-1}}\right)}{k_n} \right)} \quad (3-17)$$

For a circular pipe cross sections, this effective hZ in can now be used in [Equation 3-5](#). Note the reversed sign since Q_{wall} is the heat added to the pipe from the surroundings. The assumptions in the deduction above are:

- Equal temperature around the circumference of the pipe.
- The heat transfer through the wall is quasi static. The latter means that the wall is assumed to immediately assume the equilibrium temperature distribution corresponding to T and T_{ext} . If this assumption would not be made, an auxiliary PDE across the wall coordinate would be required.

For square and rectangular pipe shapes the average conductance can be approximated by the simpler sum of resistances across a plane wall, which can be found in for example ([Ref. 14](#)):

$$(hZ)_{\text{eff}} = \frac{1}{\frac{1}{Z_0 h_{\text{int}}} + \frac{1}{Z_N h_{\text{ext}}} + \sum_{n=1}^N \frac{w_n}{k_n ((Z_n + Z_{n-1})/2)}} \quad (3-18)$$

The film resistances can be calculated from

$$h = \text{Nu} \frac{k}{d_h} \quad (3-19)$$

where k is the thermal conductivity of the material, and Nu is the Nusselt number. d_h is the hydraulic diameter, defined as

$$d_h = 4A/Z. \quad (3-20)$$

- The inner and outer film coefficients is evaluated at $(T + T_0)/2$ and $(T_N + T_{\text{ext}})/2$, respectively.
- The thermal conductivity k_n can be temperature dependent and is evaluated at $(T_n + T_{n-1})/2$.
- To compute d_h in Equation 3-20 the local perimeter is calculated as $Z = f(w)$ and the cross section area as $A = f(w)$. Automatic calculations for circular tubes are done by the physics interface as $Z = 2\pi r$ and $A = \pi r^2$. For rectangular tubes it is $Z = 2(\text{width} + \text{height})$ and $A = \text{width} \cdot \text{height}$. For user-defined pipe shapes, the user can enter arbitrary expressions.

The local temperatures in each radial position of the pipe wall (see Figure 3-5) are computed considering the fact that Equation 3-5 also can be applied for each individual wall layer:

$$Q_{\text{wall}} = (hZ)_{\text{eff}}(T_{n+1} - T_n) \quad (3-21)$$

Combining Equation 3-5, Equation 3-21 and Equation 3-17 or Equation 3-18 (depending on pipe shape) for each wall layer explicitly gives each T_n .

Internal Film Resistance For internal laminar forced convection in fully developed pipe flow, the Nusselt number is a constant that depends on the pipe cross-section. Values are listed in the table below (Ref. 1). The Pipe Flow interface interpolates to find values for width/height ratios not listed. Default settings for film coefficient

calculations are “Automatic”, which means that laminar and turbulent correlations are applied according to the Re number.

TABLE 3-1: NUSSELT NUMBERS FOR INTERNAL LAMINAR PIPE FLOW.

CROSS SECTION	WIDTH/HEIGHT	NU
circular	-	3.66
square	1	2.98
rectangular	1.43	3.08
rectangular	2	3.39
rectangular	3	3.96
rectangular	4	4.44
rectangular	8	5.60
parallel plates	infinity	7.54

For user-defined cross sections, Nu is suggested to 3.66 as a default.

For internal turbulent forced convection ($3000 < Re < 6 \cdot 10^6$, $0.5 < Pr < 2000$), the Gnielinski equation (Ref. 19) applies:

$$Nu_{\text{int}} = \frac{(f_D/8)(Re - 1000)Pr}{1 + 12.7(f_D/8)^{1/2}(Pr^{2/3} - 1)} \quad (3-22)$$

Where Pr is the Prandtl number:

$$Pr = \frac{C_p \mu}{k} \quad (3-23)$$

The film resistance due in the internal flow can be calculated using material properties defined in the Heat Transfer feature and the calculated friction factor. Material properties are evaluated at the mean internal film temperature $(T + T_0)/2$ (see [Expressions for the Darcy Friction Factor](#)).

Using the hydraulic diameter makes the equations applicable to noncircular pipe cross sections.

External Film Resistance The material properties used should be those of the external fluid. Do not set the material to Domain Material if you have a different fluid on the inside and outside. Typically, the temperature and pressure are required to evaluate the material functions. The external fluid velocity is required for the Forced Convection option and is a user-defined input.

For External forced convection around a pipe, valid for all Re and for Pr > 0.2, the Churchill and Bernstein (Ref. 20) correlation is used:

$$\text{Nu}_{\text{ext}} = 0.3 + \frac{0.62\text{Re}^{1/2}\text{Pr}^{1/3}}{[1 + (0.4/\text{Pr})^{2/3}]^{1/4}} [1 + (\text{Re}/282,000)^{5/8}]^{4/5}. \quad (3-24)$$

For External natural convection around a pipe, the Churchill and Chu (Ref. 21) correlation is used which is recommend for Ra < 10¹²:

$$\text{Nu}_{\text{ext}} = \left\{ 0.60 + \frac{0.387\text{Ra}^{1/6}}{[1 + (0.559/\text{Pr})^{9/16}]^{8/27}} \right\}^2 \quad (3-25)$$

where the Rayleigh number is given by:

$$\text{Ra} = \text{PrGr} \quad (3-26)$$

and the Grashof number is:

$$\text{Gr} = \frac{g\beta|T_N - T_{\text{ext}}|d^3}{\left(\frac{\mu}{\rho}\right)^2} \quad (3-27)$$

Above d is the outside diameter of the pipe and, β is the fluid's coefficient of volumetric thermal expansion:

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right) \Big|_p \quad (3-28)$$

Material properties are evaluated at $(T_N + T_{\text{ext}})/2$.

Theory for the Open Pipe End Condition

At the open boundary both inflow and outflow conditions are supported. For an incoming flow, the boundary condition defines a heat flux that accounts for the energy that would normally be brought by the fluid flow if the channel upstream to the inlet was modeled.

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

$$-\mathbf{e}_t \cdot \mathbf{q} = A\rho\Delta H u I_e \quad (3-29)$$

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

$$\Delta H = \int_{T_{\text{ustr}}}^T C_p dT \quad (3-30)$$

where T_{ustr} is the upstream temperature, T is the inlet temperature, C_p is the fluid heat capacity at constant pressure.

For outgoing flow across the boundary, a zero conductive flux condition is applied:

$$-\mathbf{e}_t \cdot \mathbf{q} = 0$$

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

For large flow rates, advective heat transfer dominates over conductive heat transfer at the inlet. In this case, the left-hand side of Equation 3-29 is small compared to its right-hand side. As the heat capacity C_p is positive, in the absence of pressure contribution to the enthalpy, this induces the following constraint on temperature

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

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

For low flow rates or in the presence of large heat sources or sinks next to the inlet, the conductive heat flux cannot be neglected. The inlet temperature is adjusted to balance the energy brought by the flow at the inlet and the energy transferred by conduction from the interior.

STABILIZATION OF THE HEAT TRANSFER EQUATION

The transport equation in the Heat Transfer in Pipes interface is numerically stabilized.




Numerical Stabilization in the *COMSOL Multiphysics Reference Manual*

Reference for the Heat Transfer in Pipes Interface

1. F.P. Incropera and D.P. DeWitt, *Fundamentals of Heat and Mass Transfer*, 4th ed., John Wiley & Sons, 1996.

Chemical Species Transport Interfaces



This chapter has information about the physics interfaces found under the **Chemical Species Transport** branch ()

In this chapter:

- [The Transport of Diluted Species in Pipes Interface](#)
- [The Reacting Pipe Flow Interface](#)
- [Theory for the Transport of Diluted Species in Pipes Interface](#)

See [The Transport of Diluted Species Interface](#) in the *COMSOL Multiphysics Reference Manual* for other Chemical Species Transport interface and feature node settings.

The Transport of Diluted Species in Pipes Interface

The **Transport of Diluted Species in pipes (dsp)** interface () , found under the **Chemical Species Transport** branch () when adding a physics interface, is used to model a mass balance equation for pipes in order to compute the concentration distribution of a solute in a dilute solution, taking into account diffusion, dispersion, convection, and chemical reactions. The physics interface approximates the concentration profile in pipes and channels by a 1D assumption on lines in 2D and 3D. Depending on additional add-on products, multiple species can be modeled.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Convection and Diffusion**, **Concentration**, 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 **Transport of Diluted Species in Pipes** to select physics features from the context menu.

The **Reacting Pipe Flow Interface** node uses this physics interface and couples it automatically to Pipe Flow and Heat Transfer in pipes.

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 `dsp`.

FLUID MODEL

Select a **Fluid model** — **Newtonian** (the default), **Power law**, **Bingham**, or **Herschel-Bulkley**.

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields) — **Number of concentrations** c (SI unit: mol/m^3) and **Concentrations**.

If required, edit the name, but dependent variables must be unique within a model:



To enable multiple species (more than one concentration), you need a license for one of these modules: Battery Design Module, CFD Module, Chemical Reaction Engineering Module, Corrosion Module, Electrodeposition Module, Fuel Cell & Electrolyzer Module, Microfluidics Module, or Subsurface Flow Module.

Edge, Boundary, Point, and Pair Nodes for the Transport of Diluted Species in Pipes Interface



- [Edge, Boundary, Point, and Pair Nodes for the Transport of Diluted Species in Pipes Interface](#)
- [Theory for the Transport of Diluted Species in Pipes Interface](#)

The [Transport of Diluted Species in Pipes Interface](#) has these 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.

- [Concentration](#)
- [Convection and Diffusion](#)
- [Initial Values](#)
- [Junction](#)
- [Mass Outflow](#)
- [Open Pipe End](#)
- [Pipe Properties](#)
- [Reactions](#)
- [Wall Mass Transfer](#)



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.

Convection and Diffusion

Use the **Convection and Diffusion** node to define the fluid properties, velocity, diffusion coefficient, and dispersion model. This node is present by default.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) uses the value **From material**. For **User defined** enter a different value or expression.

The default **Dynamic viscosity** μ (SI unit: Pa·s) uses the value **From material** and describes the relationship between the shear rate and the shear stresses in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick, such as oil, have a higher viscosity.

VELOCITY

Enter a value or expression for the **Tangential velocity** u (SI unit: m^2/s). The default is $0 \text{ m}^2/\text{s}$.

DIFFUSION COEFFICIENT

Enter a value or expression for the diffusion coefficient D_c (SI unit: m^2/s). The default is $1 \times 10^{-9} \text{ m}^2/\text{s}$.

DISPERSION

Select a **Dispersion model** — **User defined** (the default), **Taylor (laminar, circular cross section)**, or **Taylor (turbulent, circular cross section)**. For **User defined**, enter a value or expression for the **Dispersion coefficient** D_D (SI unit: m^2/s). The default is $0 \text{ m}^2/\text{s}$. Find more detail in the theory section [Dispersion](#).

Initial Values

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

INITIAL VALUES

Enter values or expressions for the initial value of the **Concentration** c (SI unit: mol/m³). The default is 0 mol/m³.

Junction

The **Junction** node is used to represent mass transfer in pipe fittings with three or more connected pipes. The node supports reversible flow into each pipe branch by using conditions dependent on the flow direction. When the flow is directed into a connected pipe, an inflow concentration is prescribed. When the flow is out of the pipe, the convective-flux condition of the **Mass Outflow** node is applied instead.

For a branch b where the flow is from the junction into the pipe, the inlet concentration of species c_i is defined by summing over all branches with outflow into in the manner of

$$c_{i,b} = \frac{\sum_{k \neq b}^N \max(A_k(\mathbf{u}_k \cdot \mathbf{n}_k)c_{i,k}, 0)}{\sum_{k \neq b} \max(A_k(\mathbf{u}_k \cdot \mathbf{n}_k), 0)} \quad (4-1)$$

Here A_k is the pipe cross section area, \mathbf{u}_k is the pipe velocity, \mathbf{n}_k is the pipe outlet normal direction, $c_{i,k}$ is the concentration in pipe k , and N is the number of connected pipes. Since flow is from the junction into the branch b , this contribution is omitted in the summations. The above expression assumes that the junction volume is small enough to not introduce any hold up for the fluid or species.

Reactions

Use the **Reactions** node to account for the consumption or production of species. Define the reaction rate expression as needed, which displays on the right-hand side of the species transport equations in the **Convection and Diffusion** node.

REACTIONS

Enter a **Reaction rate** R_c (SI unit: mol/(m³·s)). The default is 0 mol/(m³·s).

Wall Mass Transfer

Use the **Wall Mass Transfer** node to model species loss to the pipe's surroundings. This could be used if the diluted species is soluble in the pipe wall or if the pipe wall is semipermeable. A mass transfer coefficient is used to model the loss rate and an external bulk concentration is specified to define the driving force.

WALL MASS TRANSFER

For each species, enter a **Mass transfer coefficient** $k_{\text{wall},c}$ (SI unit: m/s). The default is 0 m/s. For each species, select an option from the list (when available) or define the **Bulk concentration** c_{bulk} (SI unit: mol/m³). The default is 0 mol/m³.


Concentration

The **Concentration** node adds a point condition for the species concentration. For example, a $c = c_0$ condition specifies the concentration of species c .

CONCENTRATION

Specify the **Concentration** $c_{\text{in},c}$ for each species. Enter a value or expression in the field for each species. The default is 0 mol/(m³·s).

CONSTRAINT SETTINGS

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

Mass Outflow

Use the **Mass Outflow** node to define species mass outflow at a pipe end. The boundary conditions applied is

$$-\mathbf{n} \cdot A(D_i + D_{D,i})\nabla c = 0 \quad (4-2)$$

The condition omits the diffusive contribution, leaving convection as the sole contribution to transfer mass out of the pipe. Normally this constitutes a well posed condition for all outflows.

Open Pipe End



The **Open Pipe End** node is used to connect a pipe to an exterior system with known concentration. It can for example be used to represent a connection to a tank or a

reservoir. The node supports reversible flow by prescribing different conditions dependent on the flow direction at the pipe end. When the flow is directed into the pipe, the upstream inlet concentration is prescribed. When the flow is instead out of the pipe, the convective-flux condition of the [Mass Outflow](#) node is applied.

INFLOW CONCENTRATION

Specify the **Inflow concentration** $c_{in,c}$ representing the concentration in the connected tank or system. Enter a value or expression in the field for each species. The default is $0 \text{ mol}/(\text{m}^3 \cdot \text{s})$.

The Reacting Pipe Flow Interface

The **Reacting Pipe Flow (rpfl)** interface () , found under the **Chemical Species Transport** branch () when adding a physics interface, is a multiphysics interface that combines the three physics interfaces: Pipe Flow, Heat Transfer in Pipes, and Transport of Diluted Species in Pipes. The physics interface approximates the flow profiles in pipes and channels by a 1D assumption on lines in 2D and 3D. Depending on additional add-on products, multiple species can be modeled.

When this physics interface is added, these default nodes are also added to the **Model Builder** — **Fluid, Pipe Properties, Temperature, Pressure, Concentration, 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 **Reacting Pipe 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 `rpfl`.

FLUID MODEL

Select a **Fluid model** — **Newtonian** (the default), **Power law**, **Bingham**, or **Herschel-Bulkley**.

PHYSICAL MODEL

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

DEPENDENT VARIABLES

This physics interface defines these dependent variables (fields). If required, edit the name, but dependent variables must be unique within a model:

- **Pressure** p (SI unit: Pa)
- **Tangential velocity** u (SI unit: m/s)

- **Temperature** T (SI unit: K)
- **Concentration** c (SI unit: mol/m³)

DISCRETIZATION

Use this section to change the discretization order for the dependent variables.



- [The Pipe Flow Interface](#)
- [The Heat Transfer in Pipes Interface](#)
- [The Transport of Diluted Species in Pipes Interface](#)
- [Edge, Boundary, Point, and Pair Nodes for the Reacting Pipe Flow Interface](#)

Edge, Boundary, Point, and Pair Nodes for the Reacting Pipe Flow Interface

The [Reacting Pipe Flow Interface](#) has these 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.

These nodes are described in this section:

- [Initial Values](#)
- [Convection and Diffusion](#)
- [Fluid](#)
- [Fluid Properties](#)
- [Pipe Properties](#)

These nodes are described for the Pipe Flow interface (listed in alphabetical order):

- Bend
- Contraction/Expansion
- Inlet
- No Flow
- Outlet
- Phase Fractions
- The Nonisothermal Pipe Flow Interface
- Pressure
- Pump
- T-Junction
- Valve
- Volume Force

These nodes are described for the Heat Transfer in Pipes interface (listed in alphabetical order):

- External Film Resistance
- Heat Outflow
- Heat Source
- Heat Transfer
- Internal Film Resistance
- Temperature
- Wall Heat Transfer
- Wall Layer

These nodes are described for the Transport of Diluted Species in Pipes interface (listed in alphabetical order):

- Concentration
- Mass Outflow
- Reactions
- Wall Mass Transfer



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.

Pipe Properties

The **Pipe Properties** node is used to define the pipe shape and flow resistance.

SHAPE

The [Pipe Shape](#) settings are the same as for the [Pipe Properties](#) node described for the Pipe Flow interface.

FLOW RESISTANCE

The **Friction model** is **User defined** by default. Enter a value or expression for the **Darcy friction factor** f_D (dimensionless). The default is 0.

Initial Values

The **Initial Values** node adds initial values for the pressure, tangential velocity, temperature, and concentration that can serve as an initial condition for a transient simulation or as an initial guess for a nonlinear solver.

INITIAL VALUES

Enter values or expressions for the initial value of the:

- **Pressure** p (SI unit: Pa). The default is 101325 Pa.
- **Tangential velocity** u (SI unit: m/s). The default is 0 m/s.
- **Temperature** T (SI unit: K). The default is 293.15 K.
- **Concentration** c (SI unit: mol/m³). The default is 0 mol/m³.

Fluid

Use the **Fluid** node to define the density, dynamic viscosity, and heat convection and conduction properties.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m³) uses the value **From material**. For **User defined** enter a different value or expression.

The default **Dynamic viscosity** μ (SI unit: Pa·s) uses the value **From material** and describes the relationship between the shear rate and the shear stresses in a fluid. Intuitively, water and air have a low viscosity, and substances often described as thick, such as oil, have a higher viscosity.

DIFFUSION COEFFICIENT

Enter a value or expression for D_c (SI unit: m²/s). The default is 1×10^{-9} m²/s.

DISPERSION

Select a **Dispersion model** — **User defined** (the default), **Taylor (laminar, circular cross section)**, or **Taylor (turbulent, circular cross section)**. For **User defined**, enter a value or expression for the **Dispersion coefficient** D_D (SI unit: m^2/s). The default is $0 \text{ m}^2/\text{s}$. For detailed information about dispersion, see the theory section [Dispersion](#).

HEAT CONVECTION AND CONDUCTION

The default **Heat capacity at constant pressure** C_p (SI unit: $\text{J}/(\text{kg}\cdot\text{K})$), **Ratio of specific heats** γ (dimensionless), and **Thermal conductivity** k (SI unit: $\text{W}/(\text{m}\cdot\text{K})$) all use values **From material**. For **User defined** enter different values or expressions.

Fluid Properties

The **Fluid Properties** node adds the momentum 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 fluid properties of the fluid.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) is **User defined** and is $1 \times 10^3 \text{ kg}/\text{m}^3$.

The fluid model is selected as non-Newtonian, enter a value or expression for the more parameters like **Yield stress** τ_s (SI unit: Pa) and **Plastic viscosity** μ_s (SI unit: Pa·s).

Convection and Diffusion

Use the **Convection and Diffusion** node to define the fluid properties, velocity, diffusion coefficient, and dispersion model. This node is present by default.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) is **User defined** and is $1 \times 10^3 \text{ kg}/\text{m}^3$.

Enter a value or expression for the **Yield stress** τ_s (SI unit: Pa) and **Plastic viscosity** μ_s (SI unit: Pa·s). The defaults are 0 Pa and $0 \text{ Pa}\cdot\text{s}$, respectively.

VELOCITY

No user selection is required for the **Tangential velocity** u .

DIFFUSION COEFFICIENT

Enter a value or expression for the diffusion coefficient D_c (SI unit: m^2/s). The default is $1 \times 10^{-9} \text{ m}^2/\text{s}$.

DISPERSION

Select a **Dispersion model** — **User defined** (the default) or **Fan (laminar, circular cross section)**. For **User defined**, enter a value or expression for the **Dispersion coefficient** D_D (SI unit: m^2/s). The default is 0. For detailed information about dispersion, see the theory section [Dispersion](#).

Theory for the Transport of Diluted Species in Pipes Interface

The [Transport of Diluted Species in Pipes Interface](#) theory is described in this section. This physics interface solves a mass balance equation for pipes in order to compute the concentration distribution of a solute in a dilute solution, taking the flow velocity as input.

MASS CONSERVATION EQUATION

The mass transport equation for a diluted species i in an incompressible fluid flowing in a pipe is:

$$A \frac{\partial c_i}{\partial t} + A \mathbf{u} \cdot \nabla c_i = \nabla \cdot (A(D_i + D_{D,i}) \nabla c_i) + A \sum_k R_{ik} + \sum_k R_{wall, ik} \quad (4-3)$$

where A (SI unit: m^2) is the cross section area available for flow, c_i (SI unit: mol/m^3) is the diluted species concentration, and \mathbf{u} a velocity field. Further, D_i (SI unit: m^2/s) is the species diffusion coefficient and $D_{D,i}$ (SI unit: m^2/s) is the species dispersion coefficient. The second term on the right hand side, R_{ik} (SI unit: $\text{mol}/(\text{m}^3 \cdot \text{s})$), corresponds a source or sink due to chemical reaction number k for species i . Finally, $R_{wall, ik}$ (SI unit: $\text{mol}/(\text{m} \cdot \text{s})$), is a source term due to mass transfer contribution k through the pipe wall.



For information about tangential velocity in pipe flow, see [Theory for the Pipe Flow Interface](#).

Dispersion

The Transport of Diluted Species in Pipes interface can automatically calculate the axial dispersion of species transported in a solvent stream.

For laminar flow in circular straight pipes, the total dispersion is given by the sum of molecular diffusion, D_i (SI unit: m^2/s), and the effect of the velocity profile causing some fractions of an initial plane of fluid in the pipe to move faster than others. COMSOL Multiphysics uses the Taylor ([Ref. 22](#)) correlation for this second contribution, $D_{D,i}$ (SI unit: m^2/s):

$$D_{D,i} = \frac{u^2 d^2}{192 D_i} \quad (4-4)$$

This expression is valid if:

$$\frac{L}{d} > 0.04 \frac{ud}{D_i} \quad (4-5)$$

where d is the pipe diameter and L a characteristic pipe length. For turbulent conditions, Taylor (Ref. 23) suggests:

$$D_{D,i} = 10.1 \frac{d}{2} u \sqrt{\frac{f_D}{2}} \quad (4-6)$$

For non-Newtonian fluids in the laminar regime, Fan (Ref. 24) extended the analysis of Taylor:

$$D_{D,i} = k \frac{u^2 d^2}{4 D_i} \quad (4-7)$$

with k for Power-law fluids given by:

$$k = \frac{1}{2(n+3)(n+5)} \quad (4-8)$$

and k for Bingham plastic fluids:

$$k = \frac{\left(\frac{3}{8} - \frac{44}{35} \phi_0 + \frac{16}{15} \phi_0^2 + \phi_0^4 - \frac{28}{15} \phi_0^5 - \frac{3}{5} \phi_0^6 + \frac{8}{5} \phi_0^7 - \frac{29}{56} \phi_0^8 + \frac{1}{5} \phi_0^{10} - \phi_0^8 \ln \phi_0 \right)}{2(3 + 2\phi_0 + \phi_0^2)^2 (1 - \phi_0)^4} \quad (4-9)$$

The parameter ϕ_0 is

$$\phi_0 = \frac{r_0}{R} \quad (4-10)$$

where R is the pipe radius and r_0 is the radius of the plug flow region in the plastic flow, defined as

$$r_0 = \frac{2L\tau_B}{\Delta p} \quad (4-11)$$

This can be rewritten as

$$\phi_0 = \frac{4\tau_B}{d|\nabla^T p|} \quad (4-12)$$

where the tangential pressure gradient is calculated by the Pipe Flow interface.


STABILIZATION OF THE MASS TRANSFER EQUATION

The transport equation in the Transport of Diluted Species in Pipes node is numerically stabilized.



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


Acoustics Interfaces

This module has extra functionality for simulating acoustics in pipe networks. You can access this functionality under the **Acoustics** > **Pipe Acoustics** branch () when adding a physics interface. The documentation of the Pipe Acoustics interfaces is found in the *Acoustics Module User's Guide* in:

- [The Pipe Acoustics Frequency Domain and Transient Interfaces](#)
- [Theory for the Pipe Acoustics Interfaces](#)

In the *Acoustics Module User's Guide* you also find information about all the other acoustics physics interfaces as well as about the multiphysics coupling to Pressure Acoustics.

Pipe Mechanics

This module has extra functionality for computing displacements, rotations, stresses, strains in pipes using The Pipe Mechanics Interface, which you find under the **Structural Mechanics** branch () when adding a physics interface.

The documentation of [The Pipe Mechanics Interface](#) is found in the Structural Mechanics Module documentation, which is supplied with the Pipe Flow Module (irrespectively of whether a separate Structural Mechanics Module license is purchased).

Multiphysics Couplings

This chapter contains a description of all the multiphysics coupling features of the Pipe Flow Module.


Coupling Features

The Pipe Flow Module contains the following multiphysics coupling features:

- [Pipe Connection](#)
- [Pipe Connection, Diluted Species](#)
- [Pipe Wall Heat Transfer](#)
- [Acoustic–Pipe Acoustic Connection](#)
- [Structure–Pipe Connection](#)
- [Fluid–Pipe Interaction](#)

The couplings features are described below.

Pipe Connection

You can easily couple the Pipe Flow Interface with single-phase flow interfaces using the multiphysics coupling feature **Pipe Connection** () .

The **Pipe Connection** multiphysics coupling couples a 1D pipe segment modeled with the Pipe Flow interface with a 3D domain modeled by a single-phase flow interface. This coupling is available when the 3D fluid flow is simulated using laminar flow or RANS-EVM turbulent models.



This multiphysics coupling is available for 3D components.

Select **Heat Transfer** and **Heat Transfer in Pipes** interfaces to model nonisothermal pipe connection.

BOUNDARY SELECTION, FLUID

Select the boundary that corresponds to the inlet/outlet of the 3D model coupled to **Pipe Flow** interface.

POINT SELECTION, PIPE

Select the point that corresponds to the inlet/outlet of the pipe model coupled to the 3D fluid flow domain.

COUPLED INTERFACES

Select a **Fluid Flow** interface and a **Pipe Flow** interface to couple fluid flow. **Heat Transfer** and **Heat Transfer in Pipes** are optional. They can be set to None when the coupling is used to simulate isothermal flow.

CONNECTION SETTINGS

There is an automatic search for possible adjacent geometrical objects. If you want to modify the selections, select the **Manual control of selections** checkbox. If you clear it, the selections will be replaced by the automatic ones.

The connection ensure continuity in mass flux, heat flux and pressure across the connection, regardless of flow direction.




You can connect only one pipe segment to 2D surface that comprises an orifice in a 3D body



Convective Flow in a Heat Exchanger Plate: Application Library path
Pipe_Flow_Module/Tutorials/heat_exchanger_plate

Pipe Connection, Diluted Species

The **Pipe Connection, Diluted Species** () multiphysics coupling couples a 1D pipe segment modeled with the Transport of Diluted Species in Pipes interface with a 3D domain modeled by Transport of Diluted Species interface.



This multiphysics coupling is available for 3D components.

BOUNDARY SELECTION, FLUID

Select the boundary that corresponds to the inlet/outlet of the 3D model coupled to **Transport of Diluted Species in Pipes** interface.

POINT SELECTION, PIPE

Select the point that corresponds to the inlet/outlet of the pipe model coupled to the 3D domain.

COUPLED INTERFACES

Select a **Species transport** interface and a **Species transport in pipes** interface to couple species transport.

CONNECTION SETTINGS


There is an automatic search for possible adjacent geometrical objects. If you want to modify the selections, select the **Manual control of selections** checkbox. If you clear it, the selections will be replaced by the automatic ones.

The connection ensure continuity in species flux across the connection, regardless of flow direction.



You can connect only one pipe segment to 2D surface that comprises an orifice in a 3D body

Pipe Wall Heat Transfer

Add the **Pipe Wall Heat Transfer** multiphysics coupling node () to create transitions between domains modeled using the **Heat Transfer** interfaces and edges modeled using the **Heat Transfer in Pipes** or **Nonisothermal Pipe Flow** interfaces.

The **Pipe Wall Heat Transfer** adds the source term Q_{wall} (SI unit: W/m) to account for heat exchange through the pipe wall. The heat source applied on a cylinder of the pipe radius around the selected edges. The `diskavg` operator is used for source averaging around the edge.

The coupling transfers the external temperature, T_{ext} , to the **Wall Heat Transfer** feature in the pipe interface. The external temperature is obtained by performing a circumferential average of the temperature field in the **Heat Transfer** interface along a circle at the pipe radius, centered on the selected pipe edge. This averaging procedure is implemented through the `circavg` operator.



See [Built-In Operators](#) in the *COMSOL Multiphysics Reference Manual* for additional information about the `diskavg` and `circavg` operators.



[Line Heat Source](#) in the *Heat Transfer Module Manual*

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 `pwhtc1`.

EDGE SELECTION

By default, the selection of the coupling node is the edges inside the domains selected in the **Heat Transfer** interface that also selected in **Wall Heat Transfer** feature in the **Heat Transfer in Pipes** or **Nonisothermal Pipe Flow** interfaces.

COUPLED INTERFACES


This section defines the physics involved in the coupling. The **Heat transfer in pipes** and **Heat transfer** lists include all applicable physics interfaces.



Ground Heat Recovery for Radiant Floor Heating: Application Library path **Pipe_Flow_Module/Heat_Transfer/ground_heat_recovery**

Cooling of an Injection Mold: Application Library path **Pipe_Flow_Module/Heat_Transfer/mold_cooling**

Acoustic–Pipe Acoustic Connection

The **Acoustic–Pipe Acoustic Connection** multiphysics coupling () is used to couple the Pipe Acoustics interface to the Pressure Acoustics interface. The **Acoustic–Pipe Acoustic Connection** multiphysics coupling is only available with some COMSOL products (see www.comsol.com/products/specifications/).


The documentation of the [Acoustic–Pipe Acoustic Connection](#) multiphysics coupling is found in the Acoustics Module documentation.



Acoustics of a Pipe System with 3D Bend and Junction: Application Library path **Acoustics_Module/Tutorials,_Pipe_Acoustics/acoustics_pipe_system**


Probe Tube Microphone: Application Library path **Acoustics_Module/Tutorials,_Pipe_Acoustics/probe_tube_microphone**

Fluid–Pipe Interaction

The **Fluid–Pipe Interaction** () multiphysics node provides a way to transfer fluid loads from the Pipe Flow interface to the Pipe Mechanics interface.

The documentation of the [Fluid–Pipe Interaction](#) multiphysics coupling is found in the Structural Mechanics Module documentation.

Structure–Pipe Connection

The **Structure–Pipe Connection** () multiphysics node is used to couple the structural mechanics interfaces to the Pipe Mechanics interface. The **Structure–Pipe Connection** multiphysics coupling is only available with some COMSOL products (see www.comsol.com/products/specifications/).

The documentation of the [Structure–Pipe Connection](#) multiphysics coupling is found in the Structural Mechanics Module documentation.

I n d e x

- A** Application Libraries window 15
 - application library examples
 - acpr interface 124
 - nonisothermal pipe flow 45
 - pipe flow 22
 - pressure acoustics, frequency domain 124
 - water hammer 49
- B** bend (node) 31
 - bend specification 31
 - Bingham model 60
 - boundary nodes
 - heat transfer in pipes 80
 - nonisothermal pipe flow 45
 - pipe flow 22
 - reacting pipe flow 107
 - transport of diluted species in pipes 101
 - water hammer 50
- C** closed (node)
 - water hammer 52
- Colebrook's explicit formula 56
- common settings 13
- compressible flow 46
- concentration (node) 104
- contraction/expansion (node) 35
- convection and diffusion (node)
 - reacting pipe flow 110
 - transport of diluted species in pipes 102
- D** Darcy friction factor 53
 - expressions for 55
- dimensionless numbers
 - Grashof 95
 - Hedström 60, 62
 - modified Reynolds 59
 - Nusselt 93
 - Prandtl 94
 - Rayleigh 95
 - Reynolds 55
- documentation 14
- E** edge nodes
 - heat transfer in pipes 80
 - non-isothermal pipe flow 45
 - pipe flow 22
 - reacting pipe flow 107
 - transport of diluted species in pipes 101
 - water hammer 50
- emailing COMSOL 16
- extended Einstein 58
- external film resistance (node) 83
- external film resistance, theory 94
- F** flow resistance 26, 62
- fluid (node)
 - nonisothermal pipe flow 46
 - reacting pipe flow 109
- fluid models 21
- fluid properties (node) 23, 110
- forced convection 94
- G** gas-liquid, effective Reynolds number 58
- gas-liquid, friction factor multiplier 57
- Gnielinski equation for Nusselt number 94
- Grashof number 95
- H** Haaland equation 56
 - heat outflow (node) 87
 - heat source (node) 81
 - heat transfer (node)
 - heat transfer in pipes 80

- nonisothermal pipe flow 47
 - heat transfer in pipes interface 79
 - theory 88
 - Hedström number 60, 62
- I** initial values (node)
 - heat transfer in pipes 86
 - pipe flow 28
 - reacting pipe flow 109
 - transport of diluted species in pipes
 - 102
 - inlet (node) 28
 - internal film resistance (node) 83
 - internal film resistance, theory 93
 - internet resources 14
- J** junction (node)
 - transport of diluted species in pipes
 - 103
- K** knowledge base, COMSOL 16
 - Korteweg formula 75
- L** local friction loss (node) 51
 - loss coefficients 63
 - lossless fitting (node) 30
- M** mass outflow (node) 104
 - momentum and continuity equations 53
 - MPH files 15
- N** no flow (node) 30
 - nodes, common settings 13
 - nonisothermal pipe flow interface 44
 - theory 53
 - Nusselt number 93
 - n-way junction (node) 41
 - n-way junctions 63
- O** Open Pipe End (node) 104
 - outlet (node) 30
- P** pair nodes
 - heat transfer in pipes 80
 - nonisothermal pipe flow 45
 - pipe flow 22
 - reacting pipe flow 107
 - transport of diluted species in pipes
 - 101
 - water hammer 50
 - phase fractions (node) 28
 - physics interfaces, common settings 13
 - physics symbols
 - showing 42
 - pipe connection (node) 120
 - pipe cross section area 88
 - pipe flow interface 21
 - theory 53
 - pipe properties (node)
 - pipe flow 25
 - reacting pipe flow 108
 - water hammer 51
 - point nodes
 - heat transfer in pipes 80
 - nonisothermal pipe flow 45
 - pipe flow 22
 - reacting pipe flow 107
 - transport of diluted species in pipes
 - 101
 - water hammer 50
 - Prandtl number 94
 - pressure (node)
 - pipe flow 30
 - water hammer 52
 - Pressure Work 46
 - pump (node) 36
- R** Rayleigh number 95
 - reacting pipe flow interface 106
 - theory 88, 112
 - reactions (node) 103
 - Reynolds number 55
 - Reynolds number, modified 59

- S**
 - shell balance 90
 - showing
 - physics symbols 42
 - single-phase fluids 55
 - stabilization 96
 - standard settings 13
 - Stokes formula 55
 - surface roughness 27
 - values for 62
 - Swamee-Aggarwal equation 60–61
 - Swamee-Jain equation 56
 - symbols for physics 42
- T**
 - T- junction (node) 37
 - tangential velocity 54
 - technical support, COMSOL 16
 - temperature (node) 86
 - theory
 - heat transfer in pipes 88
 - non-isothermal pipe flow 53
 - pipe flow 53
 - reacting pipe flow 88, 112
 - transport of diluted species in pipes
 - 112
 - water hammer 74
 - T-junctions 63
 - transport of diluted species in pipes in-
 - terface 100
 - theory 112
 - two-phase flow 57
 - two-phase friction factor 57
- U**
 - unit tangent vector 54
- V**
 - valve (node) 31
 - velocity (node)
 - water hammer 52
 - volume force (node) 25
 - volumetric void fraction 58
 - von Kármán formula 56
- W**
 - wall heat transfer (node) 82, 86
 - wall heat transfer, theory 89
 - wall layer (node) 85
 - wall mass transfer (node) 104
 - water hammer interface 48
 - theory 74
 - websites, COMSOL 16
 - Wood equation 56
- Y**
 - Y- junction 40
 - Y-junctions 63

