

Pipe Flow Module

User's Guide

Pipe Flow Module User's Guide

© 1998–2018 COMSOL

Protected by patents listed on www.comsol.com/patents, and U.S. Patents 7,519,518; 7,596,474; 7,623,991; 8,457,932; 8,954,302; 9,098,106; 9,146,652; 9,323,503; 9,372,673; and 9,454,625. Patents pending.

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

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

Version: COMSOL 5.4

Contact Information

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

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

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

Part number: CM022801

C o n t e n t s

Chapter 1: Introduction

About the Pipe Flow Module	8
What Can the Pipe Flow Module Do?	8
The Pipe Flow Module Physics Interface Guide	9
Common Physics Interface and Feature Settings and Nodes	11
The Liquids and Gases Materials Database	11
Where Do I Access the Documentation and Application Libraries?	11
 Overview of the User's Guide	 15

Chapter 2: Fluid Flow Interfaces

Mechanisms for Modeling Fluid Flow	18
Coupling to Other Physics Interfaces	18
 The Pipe Flow Interface	 19
Edge, Boundary, Point, and Pair Nodes for the Pipe Flow Interface	20
Fluid Properties	21
Volume Force	23
Pipe Properties	23
Phase Fractions	25
Initial Values	25
Inlet.	26
Outlet.	27
No Flow	27
Lossless Fitting	28

Pressure	28
Bend	28
Valve	29
Contraction / Expansion	29
Pump	30
T-Junction	31
Y-Junction	34
n-Way Junction	36
Internal Pressure Lock.	36
Pipe Flow Physics Feature Symbols	37
Pipe Connection	37
The Nonisothermal Pipe Flow Interface	39
Edge, Boundary, Point, and Pair Nodes for the Nonisothermal Pipe Flow Interface.	40
Fluid	41
Heat Transfer	42
The Water Hammer Interface	43
Edge, Boundary, Point, and Pair Nodes for the Water Hammer Interface	44
Fluid Properties	45
Pipe Properties	45
Local Friction Loss	46
Closed.	46
Velocity	46
Pressure	46
Theory for the Pipe Flow Interface	47
Flow Equations	47
References for the Pipe Flow Interface	57

Theory for the Water Hammer Interface	60
Flow Equations	60
Reference for the Water Hammer Interface	62

Chapter 3: Heat Transfer Interfaces

The Mechanisms for Heat Transfer	64
The Heat Transfer in Pipes Interface	65
Edge, Boundary, Point, and Pair Nodes for the Heat Transfer in Pipes Interface	66
Heat Transfer	67
Heat Source	67
Wall Heat Transfer	68
Internal Film Resistance	69
External Film Resistance	70
Wall Layer	71
Initial Values	72
Temperature	72
Heat Outflow	72
Theory for the Heat Transfer in Pipes Interface	73
The Heat Transfer Equation.	73
Reference for the Heat Transfer in Pipes Interface	80

Chapter 4: Chemical Species Transport Interfaces

The Transport of Diluted Species in Pipes Interface	82
Edge, Boundary, Point, and Pair Nodes for the Transport of Diluted Species in Pipes Interface	83
Convection and Diffusion	84
Initial Values	84
Reactions.	85
Wall Mass Transfer	85
Concentration	85

Mass Outflow	86
The Reacting Pipe Flow Interface	87
Edge, Boundary, Point, and Pair Nodes for the Reacting Pipe Flow Interface	88
Pipe Properties	89
Initial Values	90
Fluid	90
Fluid Properties	91
Convection and Diffusion	91
Theory for the Transport of Diluted Species in Pipes Interface	93

Chapter 5: Acoustics Interfaces

The Pipe Acoustics Interfaces	98
The Pipe Acoustics, Frequency Domain Interface.	98
The Pipe Acoustics, Transient Interface	100
Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces	101
Initial Values	102
Fluid Properties	102
Pipe Properties	103
Closed.	104
Pressure	104
Velocity	105
End Impedance	105
Theory for the Pipe Acoustics Interfaces	108
Governing Equations	108
Theory for the Pipe Acoustics Boundary Conditions	112
Solving Transient Problems	115
Cut-off Frequency	116
Flow Profile Correction Factor β	116
References for the Pipe Acoustics Interfaces	117

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 features and Pipe Acoustics features, respectively. The Pipe Flow Module can address problems involving flow velocity, pressure, temperature 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 pipe 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. 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 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				
 Acoustic-Structure Interaction				
Pipe Acoustics, Frequency Domain ¹		pafd	3D, 2D	eigenfrequency; frequency domain
Pipe Acoustics, Transient ¹		patd	3D, 2D	time dependent
 Chemical Species Transport				
Reacting Pipe Flow		rpfl	3D, 2D	stationary; time dependent
Transport of Diluted Species in Pipes		dsp	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
 Nonisothermal Flow				
Nonisothermal Pipe Flow		nipfl	3D, 2D	stationary; time dependent
 Heat Transfer				
Heat Transfer in Pipes		htp	3D, 2D	stationary; time dependent
¹ Requires both the Pipe Flow Module and the Acoustics Module.				

Common Physics Interface and Feature Settings and Nodes

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

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



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) 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 Materials Database](#), see [Materials](#) in the *COMSOL Multiphysics Reference Manual*.

Where Do I Access the Documentation and Application Libraries?

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







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




THE DOCUMENTATION AND ONLINE HELP

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



Opening Topic-Based Help

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

	<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.• On 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.• From the File menu select Help>Documentation ().
---	---



To open the **Documentation** window:

- Press Ctrl+F1.
- On the main toolbar, click the **Documentation** () button.
- From the main menu, select **Help>Documentation**.



THE APPLICATION LIBRARIES WINDOW

Each model or application includes documentation with the theoretical background and step-by-step instructions to create a model or app. The models and applications are available in COMSOL Multiphysics as MPH files that you can open for further investigation. You can use the step-by-step instructions and the actual models as templates for your own modeling. In most models, SI units are used to describe the relevant properties, parameters, and dimensions, but other unit systems are available.

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





[The Application Libraries Window](#) in the *COMSOL Multiphysics Reference Manual*.

Opening the Application Libraries Window

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




- From the **Home** toolbar, **Windows** menu, click () **Applications Libraries**.
- From the **File** menu select **Application Libraries**.

To include the latest versions of model examples, from the **File>Help** menu, select () **Update COMSOL Application Library**.



Select **Application Libraries** from the main **File>** or **Windows>** menus.



To include the latest versions of model examples, from the **Help** menu select () **Update COMSOL Application Library**.

CONTACTING COMSOL BY EMAIL

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

COMSOL ACCESS AND TECHNICAL SUPPORT

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

COMSOL ONLINE RESOURCES

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

[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

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


THE CHEMICAL SPECIES TRANSPORT BRANCH

[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

[Acoustics Interfaces](#) chapter describes the Pipe Acoustics, Transient interface and its theory. This physics interface requires both the Acoustics Module and the Pipe Flow Module.


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 feature 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`.

FLUID MODEL

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

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

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

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

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
- Fluid Properties
- Initial Values
- Inlet
- Internal Pressure Lock
- No Flow
- n-Way Junction
- Outlet
- Phase Fractions
- The Nonisothermal Pipe Flow Interface
- Pipe Properties
- Pressure
- Pump
- T-Junction
- Valve
- Volume Force
- Y-Junction
- Lossless Fitting



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) 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.

PIPE SHAPE

Select a pipe shape from the list — **Not set** (the default), **Circular**, **Square**, **Rectangular**, 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 **User defined** enter a value or expression for the **Cross sectional area** A_c (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 Power-law and Bingham fluids in tubes of circular cross section, the **Irvine**, **Stokes**, and **Darby** friction models are available.

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).

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

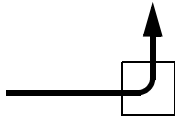
Enter a value or expression for the **Pressure** p_0 (SI unit: Pa). The default is 101325 Pa.

CONSTRAINT SETTINGS

To display this section, click the **Show** 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** can only be applied to a point which is connected to exactly two edges.

BEND SPECIFICATION

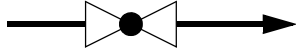
Select a **Bend** — **90 degrees standard elbow (K = 0.9)** (the default), **45 degrees standard elbow (K = 0.5)**, or **User-defined loss coefficient**. For **User-defined loss coefficient** enter a **Loss coefficient** K_f (dimensionless).

CONSTRAINT SETTINGS

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

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

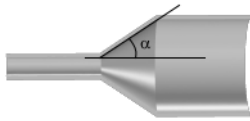
Select a **Valve**—**Globe valve (K = 10)** (the default), **Angle valve (K = 4.4)**, **Gate valve (K = 0.2)**, **Ball valve (K = 4.5)**, **Butterfly valve (K = 0.6)**, **Swing check (K = 2.5)**, or **User-defined loss coefficient**. For **User-defined loss coefficient** enter a **Loss coefficient K_f**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

CONSTRAINT SETTINGS

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

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** can only be applied to a point which is connected to exactly two pipe segments.

CONTRACTION SPECIFICATION

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

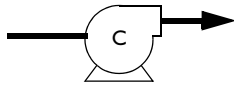
- For **Gradual**, enter an **Angle of convergence** α (SI unit: radians). The default is 0 radians.
- For **User-defined loss coefficient**, enter a **Loss coefficient** K_f (dimensionless). The default is 0.

CONSTRAINT SETTINGS

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

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 an 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.

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

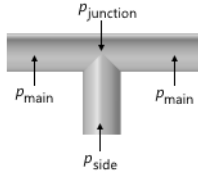


Figure 2-1: A T-junction with local pressure indications.



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](#).

FRICITION SPECIFICATION

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 respective losses are calculated as

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

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

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

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

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

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

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

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

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](#).

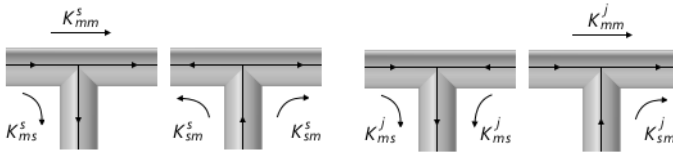


Figure 2-2: T-junction, joining and separating flows.

The losses for the outgoing branches are calculated as

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

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

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

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

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


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

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 an Y-shape.

CONSTRAINT SETTINGS

To display this section, click the **Show** 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.

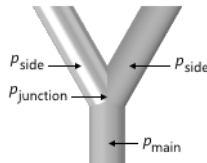


Figure 2-3: A Y-junction with local pressure indications.



The **Y-junction** can only be applied to a point that is 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 2-3](#). 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.

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 2-3](#). The respective pressure losses are calculated as

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

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

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

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

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

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

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

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 an Y-shape.

CONSTRAINT SETTINGS

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

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.

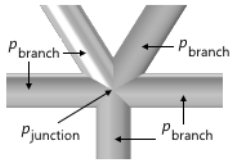


Figure 2-4: An *n*-Way Junction with local pressure indications.

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** from the **Graphics and Plot Windows** menu on [The Preferences Dialog Box](#). The check box 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

Pipe Connection

You can easily couple Pipe Flow Interface with Single-Phase Flow interfaces using multiphysics coupling feature **Pipe Connection**. The **Pipe Connection** node is only available with some COMSOL products (see <http://www.comsol.com/products/specifications/>).

The **Pipe Connection** multiphysics feature couples a 1D pipe segment modeled with the Pipe Flow interface with a 3D single phase flow body. There is an automatic search for possible adjacent geometrical objects. If you want to modify the selections, select the **Manual control of selections** check box. If you clear it, the selections will be replaced by the automatic ones.



If the automatic selection is not sufficient, select the **Manual control of selections** check box. Then, select the connected fluid boundaries in the **Boundary Selection, Fluid** section and the corresponding pipe points in the **Point Selection, Pipe** section. You can connect only one pipe segment to 2D surface that comprises an orifice in a 3D body.

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



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

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)

DISCRETIZATION

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

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](#)
- [Contraction / Expansion](#)
- [Inlet](#)
- [No Flow](#)
- [n-Way Junction](#)
- [Outlet](#)
- [Phase Fractions](#)
- [The Nonisothermal Pipe Flow Interface](#)
- [Pressure](#)
- [Pump](#)
- [T-Junction](#)
- [Valve](#)
- [Volume Force](#)
- [Y-Junction](#)



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) 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** check box 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 1·10³ 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)

DISCRETIZATION

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



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) 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 2-3](#) for links to common sections and [Table 2-4](#) 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/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.

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 the **Young's modulus** E . Select a **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 2-7](#).

FRICITION SPECIFICATION

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

CONSTRAINT SETTINGS

To display this section, click the **Show** 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** 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](#)
- [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. 16](#)):

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

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 2-5: 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 2-8](#) through [Equation 2-14](#).

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, 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:

$$\Delta p = \frac{1}{2} K_i \rho u^2 \quad (2-7)$$

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. 14](#)) and the set predefined in the Pipe Flow interface is reproduced in the table below:

TABLE 2-2: LOSS COEFFICIENTS

FEATURE	DESCRIPTION	LOSS COEFFICIENT K
Bends	90° standard elbow	0.9
	45° standard elbow	0.5
Valves	Globe, fully open	10

TABLE 2-2: LOSS COEFFICIENTS

FEATURE	DESCRIPTION	LOSS COEFFICIENT K
	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. 15](#)).

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.

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 :

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

where

$$A = \left[-2.457 \ln \left(\left(\frac{7}{\text{Re}} \right)^{0.9} + 0.27(e/d) \right) \right]^{16} \quad (2-9)$$

$$B = \left(\frac{37530}{\text{Re}} \right)^{16} \quad (2-10)$$

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

$$f_D = \frac{64}{\text{Re}} \quad (2-11)$$

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

$$f_D = 0.094(e/d)^{0.225} + 0.53(e/d) + 88(e/d)^{0.44} \cdot \text{Re}^a \quad (2-12)$$

with

$$a = -1.62(e/d)^{0.134} \quad (2-13)$$

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 ($4 \cdot 10^3 < \text{Re} < 1 \cdot 10^8$)

$$\sqrt[4]{\frac{1}{f_D}} = -1.8 \log_{10} \left(\left(\frac{e/d}{3.7} \right)^{1.11} + \left(\frac{6.9}{\text{Re}} \right) \right) \quad (2-14)$$

It can be rewritten as

$$f_D = \frac{(1/1.8)^2}{\left(\log_{10} \left(\frac{e/d}{3.7} \right)^{1.11} + \left(\frac{6.9}{\text{Re}} \right) \right)^2} \quad (2-15)$$

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

$$\sqrt[4]{\frac{1}{f_D}} = -1.8 \log_{10} \left(\frac{6.9}{\text{Re}} \right) \quad (2-16)$$

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

$$\sqrt{\frac{1}{f_D}} = -1.8 \log_{10} \left(\left(\frac{e/d}{3.7} \right)^{1.11} \right) = -2 \log_{10} \left(\frac{e/d}{3.7} \right) \quad (2-17)$$

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

$$f_D = \frac{0.25}{\left(\log_{10} \left(\left(\frac{e/d}{3.7} \right) + \frac{5.74}{(\text{Re})^{0.9}} \right) \right)^2} \quad (2-18)$$

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 Karman, or Swamee-Jain, intended for the turbulent regime, are combined with the Stokes equation for laminar flow to cover all flow conditions. When $\text{Re} < 1000$, 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 Balasubramanian and others (Ref. 25). 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 2-8 and on, such that

$$f_D = \Phi_{2P}^2 \cdot f_{D,L} \quad (2-19)$$

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. 25](#))

$$\Phi_{2P}^2 = \frac{\rho_L}{\rho} \left[1 + \omega_G \frac{(\rho_L - \rho_G)}{\rho_G} \right] \left[1 + \omega_G \frac{(\mu_L - \mu_G)}{\mu_G} \right]^{-1/4} \quad (2-20)$$

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 2-20](#) is calculated as

$$\rho = \phi_G \rho_G + (1 - \phi_G) \rho_L \quad (2-21)$$

or

$$\rho = \frac{1}{\frac{\omega_G}{\rho_G} + \frac{(1 - \omega_G)}{\rho_L}} \quad (2-22)$$

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

$$\phi_G = \frac{\rho}{\rho_G} \omega_G \quad (2-23)$$

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. 26](#)):

$$\mu = \mu_L (1 - \phi_G)^{-(\mu_L + 2.5\mu_G)/(\mu_L + \mu_G)} \quad (2-24)$$

and the other options are

Linear average

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

Reciprocal average

$$\frac{1}{\mu} = \frac{(1-\phi_G)}{\mu_L} + \frac{\phi_G}{\mu_G} \quad (2-26)$$

Liquid viscosity

$$\mu = \mu_L \quad (2-27)$$

User defined.

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

$$\mu = m \left(\frac{\partial \gamma}{\partial t} \right)^{n-1} \quad (2-28)$$

where m and n are two empirical curve fitting parameters known as the fluid consistency coefficient and the flow behavior index, respectively. 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):

$$f_D = \frac{64}{\text{Re}_{\text{MR}}} \quad (2-29)$$

$$\text{Re}_{\text{MR}} = \frac{\rho u^{(2-n)} d_h^n}{8^{(n-1)} m \left(\frac{3n+1}{4n} \right)^n} \quad (2-30)$$

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

$$f_D = 4 \left(\frac{D(n)}{\text{Re}_{\text{MR}}} \right)^{\left(\frac{1}{3n+1} \right)} \quad (2-31)$$

where

$$D(n) = \frac{2^{(n+4)}}{7^{7n}} \left(\frac{4n}{3n+1} \right)^{3n^2} \quad (2-32)$$

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

$$\text{Re}_{\text{MR}} = \frac{6464n}{(3n+1)^2} (2+n)^{\left(\frac{2+n}{1+n}\right)} \quad (2-33)$$

predicts the critical Reynolds number.

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

$$\begin{aligned} \tau &= \tau_B + \mu_B \left(\frac{\partial \gamma}{\partial t} \right) \quad \text{for } |\tau| > |\tau_B| \\ \left(\frac{\partial \gamma}{\partial t} \right) &= 0 \quad \text{for } |\tau| < |\tau_B| \end{aligned} \quad (2-34)$$

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:

$$f_{\text{D, laminar}} = \frac{64}{\text{Re}_B} + \frac{10.67 + 0.1414 \left(\frac{\text{He}}{\text{Re}_B} \right)^{1.143}}{\left(1 + 0.149 \left(\frac{\text{He}}{\text{Re}_B} \right)^{1.16} \right) \text{Re}_B} \left(\frac{\text{He}}{\text{Re}_B} \right) \quad (2-35)$$

where

$$\text{Re}_B = \frac{\rho u d_h}{\mu_B} \quad (2-36)$$

and

$$\text{He} = \frac{\rho d_h^2 \tau_B}{\mu_B} \quad (2-37)$$

is the Hedström number.

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

$$f_{\text{D, turbulent}} = 4 \cdot 10^{a_0} \text{Re}_B^{-0.193} \quad (2-38)$$

with

$$\alpha_0 = -1.47(1 + \exp(-2.9 \cdot 10^{-5} \text{He})) \quad (2-39)$$

as well as an equation covering all flow regimes:

$$f_D = (f_{D, \text{laminar}}^b + f_{D, \text{turbulent}}^b)^{\frac{1}{b}} \quad (2-40)$$

where

$$b = 1.7 + \frac{40000}{\text{Re}_B} \quad (2-41)$$



The non-Newtonian friction models outlined above do not include any effects of wall roughness. Since the laminar sub-layers 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. 12).

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. 13 and Ref. 14) are reproduced in the table below:

TABLE 2-3: 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

TABLE 2-3: SURFACE ROUGHNESS

TUBE MATERIAL	SURFACE ROUGHNESS (MM)
Wood stave	0.18–0.9
Concrete	0.3–3
Riveted steel	0.9–9
Copper and brass	0.61

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. Agarwal, “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. R.P. Chhabra and J.F. Richardson, *Non-Newtonian Flow in the Process Industries: Fundamentals and Engineering Applications*, Butterworth Heinemann, p. 111, 2004.
13. J.M. Coulson and J.F. Richardson, *Chemical Engineering, Volume. 1*, 4th ed., Pergamon Press, p. 56, 1990.
14. *Engineering and Design Liquid Process Piping*, Department of the Army, U.S. Army Corps of Engineers, Engineering Manual No.1110-1-4008, 1999.
15. 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.
16. C.L. Barnard et. al., “A Theory of Fluid Flow in Compliant Tubes”, *Biophysical Journal*, vol. 6, no. 6, pp. 717–724, 1966.
17. F.P. Incropera and D.P. DeWitt, *Fundamentals of Heat and Mass Transfer*, 4th ed., John Wiley & Sons, 1996.
18. V. Gnielinski, “New equations for heat and mass transfer in turbulent pipe and channel flow”, *Int. Chem. Eng.*, vol. 16, pp. 359–368, 1976.
19. 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.
20. 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, p. 1323–1329, 1975.
21. G. Taylor, “Dispersion of Soluble Matter in Solvent Flowing Slowly through a Tube”, *Proc. Roy. Soc. A.*, vol. 219, pp. 186–203, 1953.
22. G. Taylor, “The Dispersion of Matter in Turbulent Flow Through a Pipe”, *Proc. Roy. Soc. A.*, vol. 223, pp. 446–468, 1954.
23. 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.
24. M.V. Lurie, *Modeling of Oil Product and Gas Pipeline Transportation*, WILEY-VCH Verlag GmbH & Co., KGaA, Weinheim, 2008.

25. R. Balasubramaniam et. al., *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.

26. M. Ungarish, *Hydrodynamics of Suspensions*, Springer-Verlag, Berlin, p. 18. ISBN 3540547622 (Berlin), 0387547622 (New York), 1993.

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:

$$\frac{\partial A\rho}{\partial t} + \nabla \cdot (A\rho\mathbf{u}) = 0 \quad (2-42)$$

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 u (SI unit: m/s) is the tangential fluid velocity.

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

$$\frac{\partial A(p)\rho(p)}{\partial t} + \nabla \cdot (A(p)\rho(p)\mathbf{u}) = 0 \quad (2-43)$$

In a first order approximation, this equals

$$A_0\rho_0\left(\frac{1}{K_p} + \frac{1}{K_A}\right)\frac{\partial p}{\partial t} + \nabla \cdot (A_0\rho_0\mathbf{u}) = 0 \quad (2-44)$$

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

$$\frac{1}{c^2} = \rho_0 \left(\frac{1}{K_p} + \frac{1}{K_A} \right) \quad (2-45)$$

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 the Poisson's ration ν of the pipe's material

$$K_A = \frac{E w_{th}}{\zeta d_h} \quad (2-46)$$

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:

$$\rho \frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_D \frac{\rho}{2d_h} \mathbf{u} |\mathbf{u}| + \mathbf{F} \quad (2-47)$$

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 $\mathbf{F} = \rho \mathbf{g}$)

$$A \frac{1}{c^2} \frac{\partial p}{\partial t} + \nabla \cdot (A \rho \mathbf{u}) = 0 \quad (2-48)$$

$$\rho \frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_D \frac{\rho}{2d_h} \mathbf{u} |\mathbf{u}| + \rho \mathbf{g} \quad (2-49)$$


BOUNDARY CONDITIONS

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

Reference for the Water Hammer Interface

I. M. Ghidaoui et al., “A Review of Water Hammer Theory and Practice”, *Appl. Mech. Rev.* 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 [The Heat Transfer Interfaces](#) 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 [The Heat Transfer Interfaces](#) 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 groundflow 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. It provides 1D models to define the pipe flow profile and 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

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



- [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](#)



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) 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: $\text{J}/(\text{kg}\cdot\text{K})$), **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.

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: $W/(m^3 \cdot K)$).



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.



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

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.

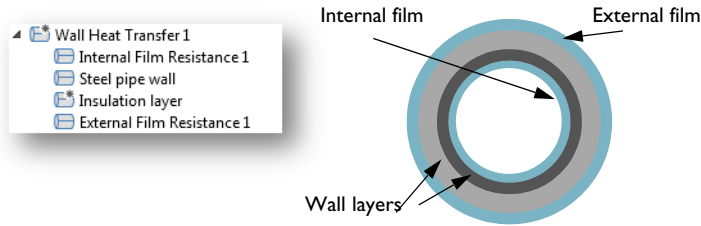


Figure 3-1: Internal and external films and the associated film coefficients (blue), and wall layers (dark and light gray).

HEAT TRANSFER MODEL

Select an option from the **External temperature** list—**User defined**, or **Temperature (ht/ht)**, the latter representing a temperature field calculated by a surrounding 3D solid physics. For **User defined** 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 check box **Take average in adjacent domain** should be cleared.

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, **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, **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](#)) require a temperature dependent density, or else the coefficient of thermal expansion β evaluates to 0 ([Equation 3-26](#)), 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.

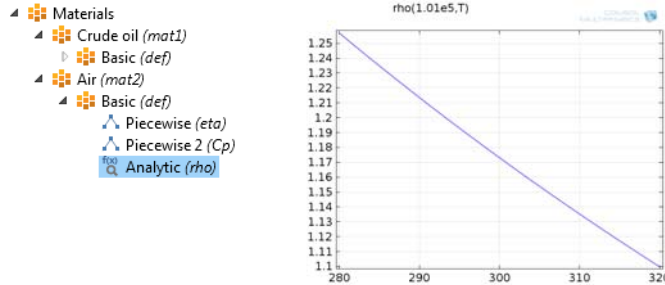


Figure 3-2: Before you use the Natural Convection option, it is good practice to select Density in the used material, and click Plot in the Settings window. That way you can inspect the behavior and make sure it is smooth within the temperature interval you plan to use it.

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, **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.

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** button () and select **Advanced Physics Options**.

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.

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. 24](#)):

$$\rho A C_p \frac{\partial T}{\partial t} + \rho A C_p \mathbf{u} \cdot \nabla T = \nabla \cdot A k \nabla T + f_D \frac{\rho A}{2d_h} |\mathbf{u}|^3 + Q + Q_{\text{wall}} + Q_p \quad (3-1)$$

where ρ is the fluid density (SI unit: kg/m³), A is the pipe cross section area (SI unit: m²) available for flow, C_p (SI unit: J/(kg·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: W/(m·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](#) check box:

$$Q_p = -\frac{TA}{\rho} \left(\frac{\partial \rho}{\partial T} \right) \left(\frac{\partial p}{\partial T} + \mathbf{u} \cdot \nabla p \right) \quad (3-2)$$

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

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

In [Equation 3-3](#), $(hZ)_{\text{eff}}$ 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 3-3](#). Q_{wall} appears as a source term in the pipe heat transfer equation, [Equation 3-1](#).

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 3-3](#) can be a constant, parameter, expression, or given by a temperature field computed by another physics interface, typically a 3D Heat Transfer interface. h is automatically calculated through film resistances and wall layers that are added as subnodes; see [Equation 3-17](#) on.

If T_{ext} is given as the temperature field computed by another 3D Heat Transfer interface, automatic heat transfer coupling is done to the 3D physics side as a line source. 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 3-3](#)) 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 3-3](#).

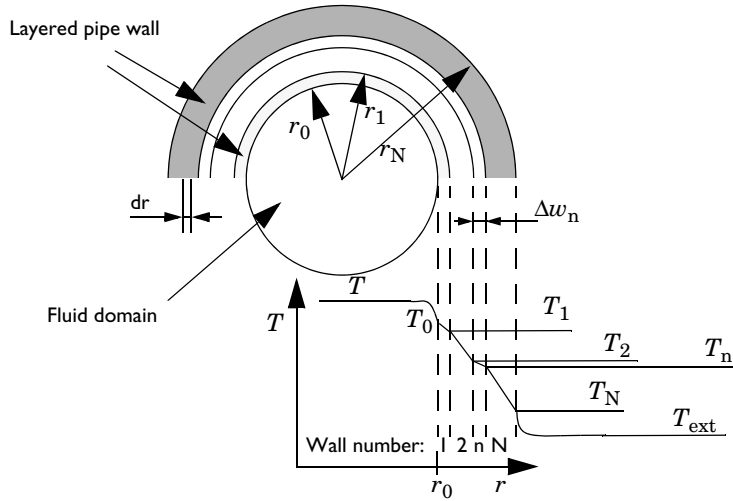


Figure 3-3: Temperature distribution across the pipe wall.

- 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: $\text{W}/(\text{m}^2 \cdot \text{K})$).
- k_n is the thermal conductivity (SI unit: $\text{W}/(\text{m} \cdot \text{K})$) of wall n

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

$$Q_0 = h_{\text{int}} A_Q \cdot (T - T_0) \quad (\text{W}) \quad (3-4)$$

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).

$$Q_1 = \Delta L 2\pi r \cdot \left(-k_1 \frac{dT}{dr} \right) \quad (3-5)$$

Rearrange and integrate from r_0 to r_1 .

$$\int_{T_0}^{T_1} dT = - \int_{r_0}^{r_1} \left(\frac{Q_1}{\Delta L 2\pi k_1} \cdot \frac{1}{r} \right) dr \quad (3-6)$$

Perform the integration

$$T_1 - T_0 = - \frac{Q_1}{\Delta L 2\pi k_1} \cdot \ln\left(\frac{r_1}{r_0}\right) \quad (3-7)$$

and rearrange

$$Q_1 = \frac{\Delta L 2\pi k_1}{\ln\left(\frac{r_1}{r_0}\right)} \cdot (T_0 - T_1) \quad (3-8)$$

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 $Q_n = Q$.

$$\begin{aligned} Q &= h_{\text{int}} \Delta L 2\pi r_0 \cdot (T - T_0) \\ Q &= \frac{\Delta L 2\pi k_1}{\ln(r_1/r_0)} \cdot (T_0 - T_1) \\ Q &= \frac{\Delta L 2\pi k_2}{\ln(r_2/r_1)} \cdot (T_1 - T_2) \\ Q &= h_{\text{ext}} \Delta L 2\pi r_2 \cdot (T_2 - T_{\text{ext}}) \end{aligned} \quad (3-9)$$

Substituting

$$\Theta = \Delta L 2\pi, \quad (3-10)$$

and making a linear combination of the equations [Equation 3-9](#) gives

$$Q = (hA_Q)_{\text{eff}} \cdot (T - T_{\text{ext}}) \quad (3-11)$$

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

$$(hA_Q)_{\text{eff}} = \frac{1}{\frac{1}{r_0 h_{\text{int}} \Theta} + \frac{\ln(r_1/r_0)}{k_1 \Theta} + \frac{\ln(r_2/r_1)}{k_2 \Theta} + \frac{1}{r_2 h_{\text{ext}} \Theta}} \quad (3-12)$$

For the general case with N wall layers this reads

$$(hA_Q)_{\text{eff}} = \frac{1}{\frac{1}{r_0 h_{\text{int}} \Theta} + \sum_{n=1}^N \left(\frac{\ln\left(\frac{r_n}{r_{n-1}}\right)}{k_n \Theta}\right) + \frac{1}{r_N h_{\text{ext}} \Theta}} \quad (3-13)$$

Now let

$$(hZ)_{\text{eff}} = \frac{(hA_Q)_{\text{eff}}}{\Delta L}, \quad (3-14)$$

where Z (SI unit: m) is an average perimeter (circumference), of the pipe, taken over the thickness of the pipe walls. Combine [Equation 3-10](#) and [Equation 3-14](#) such that

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

and insert in [Equation 3-13](#):

$$(hZ)_{\text{eff}} = \frac{2\pi}{\frac{1}{r_0 h_{\text{int}}} + \frac{1}{r_N h_{\text{ext}}} + \sum_{n=1}^N \left(\frac{\ln\left(\frac{r_n}{r_{n-1}}\right)}{k_n}\right)} \quad (3-15)$$

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

- equal temperature around the circumference of the
- 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. 13](#)):

$$(hZ)_{\text{eff}} = \frac{1}{\frac{1}{Z_0 h_{\text{int}}} + \frac{1}{Z_N h_{\text{ext}}} + \sum_{n=1}^N \frac{w_n}{k_n ((Z_n + Z_{n-1})/2)}} \quad (3-16)$$

The film resistances can be calculated from

$$h = \text{Nu} \frac{k}{d_h} \quad (3-17)$$

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

$$d_h = 4A/Z. \quad (3-18)$$

- 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 3-18 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 3-3) are computed considering the fact that Equation 3-3 also can be applied for each individual wall layer:

$$Q_{\text{wall}} = (hZ)_{\text{eff}}(T_{n+1} - T_n) \quad (3-19)$$

Combining Equation 3-3, Equation 3-19 and Equation 3-15 or Equation 3-16 (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: NUSSULT NUMBERS FOR INTERNAL LAMINAR PIPE FLOW

CROSS SECTION	WIDTH/HEIGHT	NU
circular	-	3.66
square	1	2.98
rectangular	1.43	3.08

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

CROSS SECTION	WIDTH/HEIGHT	NU
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 ($3000 < \text{Re} < 6 \cdot 10^6$, $0.5 < \text{Pr} < 2000$), the Gnielinski equation (Ref. 18) applies:

$$\text{Nu}_{\text{int}} = \frac{(f_D/8)(\text{Re} - 1000)\text{Pr}}{1 + 12.7(f_D/8)^{1/2}(\text{Pr}^{2/3} - 1)} \quad (3-20)$$

Where Pr is the Prandtl number:

$$\text{Pr} = \frac{C_p \mu}{k} \quad (3-21)$$

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).

The using the hydraulic diameter makes the equations applicable to non-circular 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 $\text{Pr} > 0.2$, the Churchill and Bernstein (Ref. 19) correlation is used:

$$\text{Nu}_{\text{ext}} = 0.3 + \frac{0.62 \text{Re}^{1/2} \text{Pr}^{1/3}}{[1 + (0.4/\text{Pr})^{2/3}]^{1/4}} [1 + (\text{Re}/282000)^{5/8}]^{4/5}. \quad (3-22)$$

For External natural convection around a pipe, the Churchill and Chu (Ref. 20) correlation is used which is recommend for $Ra < 10^{12}$:

$$Nu_{\text{ext}} = \left\{ 0.60 + \frac{0.387Ra^{1/6}}{[1 + (0.559/Pr)^{9/16}]^{8/27}} \right\}^2 \quad (3-23)$$

where the Rayleigh number is given by:

$$Ra = PrGr \quad (3-24)$$

and the Grashof number is:

$$Gr = \frac{g\beta|T_N - T_{\text{ext}}|d^3}{\left(\frac{\mu}{\rho}\right)^2} \quad (3-25)$$

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

$$\beta = -\frac{1}{\rho} \left(\frac{\partial \rho}{\partial T} \right) \bigg|_p \quad (3-26)$$

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

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

I. 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**, or **Bingham**.

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: Batteries & Fuel Cells Module, CFD Module, Chemical Reaction Engineering Module, Corrosion Module, Electrodeposition Module, Microfluidics Module, or the Subsurface Flow Module.

DISCRETIZATION

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



- [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](#)

Edge, Boundary, Point, and Pair Nodes 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](#)
- [Mass Outflow](#)
- [Pipe Properties](#)
- [Reactions](#)
- [Wall Mass Transfer](#)



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) 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/m^3). The default is $0 \text{ mol}/\text{m}^3$.

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: $\text{mol}/(\text{m}^3 \cdot \text{s})$). The default is $0 \text{ mol}/(\text{m}^3 \cdot \text{s})$.

Wall Mass Transfer

Use the **Wall Mass Transfer** node to model species loss to the surroundings of the pipe. 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 external bulk concentration is specified to define the driving force.

WALL MASS TRANSFER

For each species, enter a **Mass transfer coefficient** $k_{wall,c}$ (SI unit: m/s). The default is $0 \text{ m}/\text{s}$. For each species, select an option from the list (when available) or define the **Bulk concentration** c_{bulk} (SI unit: mol/m^3). The default is $0 \text{ mol}/\text{m}^3$.


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_{inc} (SI unit: $\text{mol}/(\text{m}^3 \cdot \text{s})$) for each species. Enter a value or expression in the field for each species. The default is $0 \text{ mol}/(\text{m}^3 \cdot \text{s})$.



CONSTRAINT SETTINGS

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

Mass Outflow

Use the **Mass Outflow** node to define species mass outflow. Mathematically this is defined so that convection is the sole contribution to species mass transfer. Normally this constitutes a well posed condition for all outflows.

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**, or **Bingham**.

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

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



- [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 2-3](#) for links to common sections and [Table 2-4](#) 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 Pa·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:

$$A \frac{\partial c_i}{\partial t} + A \mathbf{u} \cdot \nabla c_i = \nabla \cdot (A(D_i + D_{D,i})\nabla c_i) + A \sum_k R_{ik} + \sum_k R_{wall,ik} \quad (4-1)$$

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 to 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. 21) correlation for this second contribution, $D_{D,i}$ (SI unit: m^2/s):

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

This expression is valid if:

$$\frac{L}{d} > 0.04 \frac{ud}{D_i} \quad (4-3)$$

where d is the pipe diameter and L a characteristic pipe length. For turbulent conditions, Taylor (Ref. 22) suggests:

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

For non-Newtonian fluids in the laminar regime, Fan (Ref. 23) extended the analysis of Taylor:

$$D_{D,i} = k \frac{u^2 d^2}{4 D_i} \quad (4-5)$$

with k for Power-law fluids given by:

$$k = \frac{1}{2(n+3)(n+5)} \quad (4-6)$$

and k for Bingham plastic fluids:

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

The parameter ϕ_0 is

$$\phi_0 = \frac{r_0}{R} \quad (4-8)$$

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

$$r_0 = \frac{2L\tau_B}{\Delta p} \quad (4-9)$$

This can be rewritten as

$$\phi_0 = \frac{4\tau_B}{d|\nabla^T p|} \quad (4-10)$$

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, which is found under the **Acoustics>Acoustic-Structure Interaction** branch () when adding a physics interface.

- [The Pipe Acoustics Interfaces](#)
- [Theory for the Pipe Acoustics Interfaces](#)

See the *Acoustics Module User's Guide* for information about all other Acoustics physics interfaces.

If you do not have the Acoustics Module, see [The Pressure Acoustics, Frequency Domain Interface](#) in the *COMSOL Multiphysics Reference Manual* for an Acoustic physics interface and its feature node settings.

The Pipe Acoustics Interfaces



In this section:

- [The Pipe Acoustics, Frequency Domain Interface](#)
- [The Pipe Acoustics, Transient Interface](#)
- For links to all the physics features, go to [Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces](#)



These interfaces require both the Pipe Flow Module and the Acoustics Module.

The Pipe Acoustics, Frequency Domain Interface

The **Pipe Acoustics, Frequency Domain (pafd)** interface () , found under the **Acoustics>Acoustic-Structure Interaction** branch () when adding a physics interface, is used to compute the acoustic pressure and velocity variations when modeling the propagation of sound waves in flexible pipe systems. The governing equations are formulated in a general way to include the possibility of a stationary background flow. The physics interface can for example be used to compute the propagation of sound waves in HVAC systems, other large piping systems, or simply in an organ pipe.

In the frequency domain all sources and variations are assumed to be harmonic. The solved equations assume that the propagating waves are plane. The propagation of higher-order modes that exist above their cut-off frequency, dictated by the pipe cross section, is not modeled.

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduce the equations to a 1D component with scalar dependent variables. The physics interface is available in 3D on edges and 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 and point conditions. You can also right-click **Pipe Acoustics, Frequency Domain** 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 `pafd`.

SOUND PRESSURE LEVEL SETTINGS

The zero level on the dB scale varies with the type of fluid. That value is a reference pressure that corresponds to 0 dB. This variable occurs in calculations of the sound pressure level L_p based on the root mean square (rms) pressure p_{rms} , such that

$$L_p = 20 \log \left(\frac{p_{\text{rms}}}{p_{\text{ref}}} \right) \quad \text{with} \quad p_{\text{rms}} = \sqrt{\frac{1}{2} p p^*}$$

where p_{ref} is the reference pressure and the star (*) represents the complex conjugate. This is an expression valid for the case of harmonically time-varying acoustic pressure p .

Select a **Reference pressure for the sound pressure level** based on the fluid type:

- **Use reference pressure for air** to use a reference pressure of 20 μPa ($20 \cdot 10^{-6}$ Pa).
- **Use reference pressure for water** to use a reference pressure of 1 μPa ($1 \cdot 10^{-6}$ Pa).
- **User-defined reference pressure** to enter a reference pressure $p_{\text{ref, SPL}}$ (SI unit: Pa).

The default value is the same as for air, 20 μPa .



For postprocessing, plot the sound pressure level `pafd.Lp`, which depends on the selected reference pressure. You can also plot the rms intensity magnitude `pafd.I_rms` or the instantaneous intensity magnitude `pafd.I_inst`.

DEPENDENT VARIABLES

This section is used to define the dependent variables (fields) for **Pressure** p (SI unit: Pa) and **Tangential velocity** u (SI unit: m/s). If required, edit the name, but dependent variables must be unique within a model.



DISCRETIZATION

This section controls the element types used in the finite element formulation.



- [Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces](#)
- [Theory for the Pipe Acoustics Interfaces](#)

The Pipe Acoustics, Transient Interface

The **Pipe Acoustics, Transient (patd)** interface () , found under the **Acoustics>Acoustic-Structure Interaction** branch () when adding a physics interface, is used to compute the acoustic pressure and velocity variations when modeling the propagation of sound waves in flexible pipe systems. The governing equations are formulated in a general way to include the possibility of a stationary background flow. The physics interface can for example be used to compute the propagation of sound waves in HVAC systems, other large piping systems, or simply in an organ pipe.

The solved equations assume that the propagating waves are plane. The propagation of higher-order modes that exist above their cut-off frequency, dictated by the pipe cross section, is not modeled.

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduce the equations to a 1D component with scalar dependent variables. The physics interface is available in 3D on edges and 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 and point conditions. You can also right-click **Pipe Acoustics, Transient** 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 `patd`.

TRANSIENT SOLVER SETTINGS

Select the **Time stepping** (method) as **Manual** (default and recommended) or **Automatic/free** and then enter the **Maximum frequency to resolve** in the model. The default frequency is set to 1000[Hz] but should be changed to reflect the frequency content of the sources used in the model. The generated solver will be adequate in most situations if the computational mesh also resolves the frequency content in the model. Note that any changes made to these settings (after the model is solved the first time) will only be reflected in the solver if **Show Default Solver** or **Reset Solver to Defaults** is selected in the study.

The rest of the settings are the same as for [The Pipe Acoustics, Frequency Domain Interface](#).



- [Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces](#)
 - [Theory for the Pipe Acoustics Interfaces](#)
-

Edge, Boundary, Point, and Pair Nodes for the Pipe Acoustics Interfaces

[The Pipe Acoustics, Frequency Domain Interface](#) and [The Pipe Acoustics, Transient Interface](#) have these edge, boundary, point, and pair nodes, listed in alphabetical order, available from the **Physics** ribbon toolbar (Windows users), **Physics** context menu (Mac or Linux users), or right-click to access the context menu (all users).



In general, to add a node, go to the **Physics** toolbar, no matter what operating system you are using. Subnodes are available by clicking the parent node and selecting it from the **Attributes** menu.

- Closed
- End Impedance
- Fluid Properties
- Initial Values
- Pipe Properties
- Pressure
- Velocity
- Volume Force is described for the Pipe Flow interface.



In the *COMSOL Multiphysics Reference Manual* see [Table 2-3](#) for links to common sections and [Table 2-4](#) 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.

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).

Fluid Properties

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

MODEL INPUTS

Enter a value for the and **Background mean flow pressure** p_0 (SI unit: Pa). This pressure will also serve as input the material parameters that depend on the pressure.

BACKGROUND VELOCITY

Enter a value or expression for the **Background mean flow velocity** u_0 (SI unit: m/s)

Physically sound background variables for the pressure p_0 and velocity u_0 can be obtained by solving a Pipe Flow model on the same geometry.

PHYSICAL PROPERTIES

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

The default **Density** ρ (SI unit: kg/m^3) and **Speed of sound** c_s (SI unit: m/s) use the values **From material**. For **User defined** enter different values or expressions.

Pipe Properties

The **Pipe Properties** node is used to define the pipe shape, pipe model, wall drag force, and flow profile correction factor.

PIPE SHAPE

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

- For **Circular** enter a value or expression for the **Inner diameter** d_i (SI unit: m). The default is 10 cm (0.01 m).
- For **Square** enter a value or expression for the **Inner width** w_i (SI unit: m). The default is 5 cm (0.005 m).
- 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 **User defined** enter a value or expression for the **Cross sectional area** A (SI unit: m^2 ; the default is 0.01 m^2) and **Wetted perimeter** Z (SI unit: m ; the default is 0.4 m).

PIPE MODEL

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

When **Zero axial stress**, **Anchored at one end**, or **Anchored at both ends** is chosen, select an option from the **Young's modulus** E (SI unit: Pa) and **Wall thickness** Δw lists — **Not set** (the default) or **User defined**. For **User defined** in either case, enter different values or expressions.

For **Anchored at one end** or **Anchored at both ends** also select an option from the **Poisson's ratio** ν (dimensionless) list — **Not set** (the default) or **User defined**. For **User defined** enter a value or expression for Poisson's ratio.

WALL DRAG FORCE

Enter a value or expression for τ_w (SI unit: N/m^2). The default is $0 \text{ N}/\text{m}^2$.

FLOW PROFILE CORRECTION FACTOR

Enter a value or expression for β (dimensionless). The default is 1. For most practical applications this correction factor is 1 as the propagating waves are assumed plane and uniform. This value should typically be changed if a wall drag force is introduced or if a non-plug-flow background flow field is used. The flow profile correction factor is defined as

$$\beta = \left(\int (\tilde{\mathbf{u}} + \tilde{\mathbf{u}}_0)^2 dA \right) / \left(\int (\tilde{\mathbf{u}} + \tilde{\mathbf{u}}_0) dA \right)^2$$

where the velocity field is the actual one that exists in the pipe cross section. The factor thus measures the deviation from a flat background flow profile (plug flow) and a plane propagating acoustic wave.



Flow Profile Correction Factor β

Closed

Use the **Closed** node to impose zero velocity. This is the default condition added on all end points.



Theory for the Pipe Acoustics Boundary Conditions

Pressure

Use the **Pressure** node to define the boundary pressure at the pipe ends.

PRESSURE


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

In the frequency domain p represents the amplitude and phase (if it is complex valued) of a harmonic pressure source.

In the time domain enter an expression for the pressure p , for example, a forward moving sinusoidal wave of amplitude 1 Pa can be written as

$1[\text{Pa}] * \sin(\omega * t - k * x)$, where ω and k are parameters defining the angular frequency and wave number, respectively.

CONSTRAINT SETTINGS

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



Theory for the Pipe Acoustics Boundary Conditions

Velocity

Use the **Velocity** node to prescribe a velocity at the pipe ends.

VELOCITY

Enter a value or expression for the velocity u_{in} (SI unit: m/s) at the inlet and/or outlet of a pipe. The default is 0 m/s. The velocity u_{in} is defined relative to background flow u_0 and thus in the tangential coordinate system. Enable the **Show physics symbols** from the **Graphics and Plot Windows** menu on the Preference dialog box in order to visualize the boundary or edge tangent direction. Click the **Fluid Properties** node to see the tangents as a red arrows.



- [Theory for the Pipe Acoustics Interfaces](#)
- [The Preferences Dialog Box](#) in the *COMSOL Multiphysics Reference Guide*

End Impedance

Use the **End Impedance** node to model conditions at the end of a pipe. The condition can either model an infinite pipe and thus represent the characteristic impedance of the pipe system at that point. This results in a zero reflection condition. Alternatively, the condition can represent the radiation impedance of an open pipe in either a flanged (in an infinite baffle) or unflanged (a pipe ending in free open space). The end impedance

can also be user defined and could then represent modeled or experimental values for a specific pipe configuration.



The wave speed c in the pipe can be different from the speed of sound c_s in an open space. It then depends on the elastic properties of the pipe structure. It is defined in Equation 5-4 in the [Governing Equations](#) section.

The wave speed can be evaluated as $\text{sqrt}(1/\text{patd.invc2})$ or $\text{sqrt}(1/\text{pafd.invc2})$ during the analysis and results stage.

END IMPEDANCE

Select an **Impedance model**.

For [The Pipe Acoustics, Transient Interface](#) select **Infinite pipe (low Mach number limit)** (the default) or **User defined**. The **Infinite pipe (low Mach number limit)** models an infinite pipe by specifying the characteristic impedance at that point. This condition creates a non-reflecting boundary. The expression is valid for small values of the Mach number $\text{Ma} = u_0/c$. For **User defined** enter an **End impedance** Z_{end} (SI unit: Pa·s/m). The default is $\text{patd.rho} * (\text{sqrt}(1/\text{patd.invc2}))$, which is $\rho \cdot c$.

For [The Pipe Acoustics, Frequency Domain Interface](#) select from the following — **Infinite pipe (low Mach number limit)** (the default), **Infinite pipe, Flanged pipe, circular, Flanged pipe, rectangular, Unflanged pipe, circular (low ka limit), Unflanged pipe, circular**, or **User defined**.

- For **Infinite pipe** enter a **Wave number** k (SI unit: rad/m). The default expression is $\text{pafd.omega} * (\text{sqrt}(\text{pafd.invc2}))$. This end impedance models the infinite pipe using the full (nonlinear) dispersion relation. It is valid for all Mach numbers but requires the additional input of the wave number k .
- For **Flanged pipe, circular** enter an **Inner radius** a (SI unit: m). The default expression is $\text{pafd.dh}/2$. This end impedance models the radiation impedance of a circular pipe terminated in an infinite baffle. It is an exact analytical result valid for all frequencies and pipe radii. In the low frequency limit it reduces to the classical results:

$$Z_{\text{end}} = \rho c \left(\frac{1}{2} (ka)^2 + i(0.8216 \cdot ka) \right)$$

- For **Flanged pipe, rectangular** enter an **Inner width** w_i (SI unit: m). The default is 5 cm (0.005 m). Also enter an **Inner height** h_i (SI unit: m). The default is 10 cm (0.01 m).

This end impedance models the radiation impedance of a pipe of rectangular cross section terminated in an infinite baffle. The model is only valid in the low frequency range where $kw_i \ll 1$ and $kh_i \ll 1$.

- For **Unflanged pipe, circular (low ka limit)** or **Unflanged pipe, circular** enter an **Inner radius** a (SI unit: m). The default expression is `paFd.dh/2`. These two end impedance models prescribe the radiation impedance of an unflanged circular pipe (a pipe ending in free open space). The first model is the classical low-frequency approximation valid for $ka \ll 1$. While the second model extends the frequency range to $ka < 3.83$.
- For **User defined** enter an **End impedance** Z_{end} (SI unit: Pa·s/m). The default expression is `paFd.rho*(sqrt(1/paFd.invc2))`, which is $\rho \cdot c$.



For a detailed review of the end impedance models see: [Theory for the Pipe Acoustics Boundary Conditions](#).

Theory for the Pipe Acoustics Interfaces

The equations governing the propagation of sound in pipes stem from considering momentum, mass, and energy balances for a control volume of a piece of pipe. The resulting equations are expressed in the cross-sectional averaged variables and reduces the equations to a 1D component with scalar dependent variables. The present theory assumes no thermal conduction and thus no losses due to thermal conduction (isentropic sound propagation). The Pipe Acoustics, Transient and the Pipe Acoustics, Frequency Domain interfaces require both the Pipe Flow Module and the Acoustics Module.

In this section:

- [Governing Equations](#)
- [Theory for the Pipe Acoustics Boundary Conditions](#)
- [Solving Transient Problems](#)
- [Cut-off Frequency](#)
- [Flow Profile Correction Factor \$\beta\$](#)
- [References for the Pipe Acoustics Interfaces](#)

Governing Equations

The continuity equation derived for a control volume is given by

$$\frac{\partial(A\rho)}{\partial t} + \nabla \cdot (A\rho\mathbf{u}) = 0 \quad (5-1)$$

and the corresponding momentum balance equation is

$$\frac{\partial(\rho A\mathbf{u})}{\partial t} + \nabla(\rho A\beta\mathbf{u}^2) = -A\nabla p + \tau_w Z + A\mathbf{F} \quad (5-2)$$

where Z is the inner circumference of the pipe and $A = A(x, p, \dots)$ is the inner wetted cross-sectional area, \mathbf{u} is the area-averaged mean velocity, which is also defined in the tangential direction $\mathbf{u} = u\mathbf{e}_t$, p is the mean pressure along the pipe, τ_w is the wall drag force, and \mathbf{F} is a volume force. The gradient is taken in the tangential direction \mathbf{e}_t . The

term β is a flow profile correction factor relating the mean of the squared total velocity to the square of the mean velocity. Such that

$$u = \frac{1}{A} \int \tilde{\mathbf{u}} \cdot d\mathbf{A} \quad p = \frac{1}{A} \int \tilde{p} dA \quad \beta = \left(\frac{1}{A} \left(\int \tilde{u}^2 dA \right) \right) / u^2 \quad (5-3)$$

where

$$\tilde{p} = \tilde{p}(\mathbf{x}) \quad \text{and} \quad \tilde{u} = \tilde{u}(\mathbf{x})$$

are the local non-averaged parameters. Again p and u are the area-averaged dependent variables.

LINEARIZATION

The governing equations are now linearized; that is, all variables are expanded to first order assuming stationary zero-order values (steady-state background properties). The acoustic variations of the dependent variables are assumed small and on top of the background values. This is done according to the following scheme:

$$\begin{aligned} \mathbf{u}(x, t) &= \mathbf{u}_0(x) + \mathbf{u}_1(x, t) \\ p(x, t) &= p_0(x) + p_1(x, t) \\ \rho(x, t) &= \rho_0(x) + \rho_1(x, t) \\ A(x, t) &= A_0(x) + A_1(x, t) \end{aligned}$$

where A_0 is often only function of \mathbf{x} ; however, A_0 can be changed by external factors such as heating or structural deformation, thus the time dependency. The first-order terms represent small perturbations on top of the background values (zero order). They are valid for

$$\rho_1 \ll \rho_0 \quad p_1 \ll \rho_0 c_0^2 \quad |\mathbf{u}_1| \ll c_0 \quad A_1 \ll A_0$$

Moreover, the perturbations for the fluid density and cross-sectional area are expanded to first order in p_0 in a Taylor series such that

$$\begin{aligned} \rho_1 &= \rho - \rho_0 = (p - p_0) \left[\frac{\partial \rho}{\partial p} \right]_s \Big|_0 \\ A_1 &= A - A_0 = (p - p_0) \left[\frac{\partial A}{\partial p} \right]_s \Big|_0 \end{aligned}$$

where the subscript s refers to constant entropy; that is, the processes are isentropic. The relations for the fluid compressibility and the cross-sectional area compressibility are

$$\beta_0 = \frac{1}{K_0} = \frac{1}{K_s} = \frac{1}{\rho_0} \left[\frac{\partial \rho}{\partial p} \right]_s \bigg|_0 = \frac{1}{\rho_0 c_s^2}$$

$$\beta_A = \frac{1}{A_0} \left[\frac{\partial A}{\partial p} \right]_s \bigg|_0 = \frac{1}{K_A}$$

Here, β_0 is the fluid compressibility at the given reference pressure p_0 , the isentropic bulk speed of sound is denoted c_s , and ρ_0 is the fluid density at the given reference temperature and reference pressure. β_A is the effective compressibility of the pipe's cross-sectional A_0 due to changes in the inner fluid pressure. The bulk modulus K is equal to one over the compressibility.

Inserting the above expansions into the governing equations (Equation 5-1 and Equation 5-2) and retaining only first-order terms yield the pipe acoustics equations including background flow. These are:

$$A_0 \frac{1}{c^2} \frac{\partial p_1}{\partial t} + \nabla \cdot \left(A_0 \rho_0 \left(\mathbf{u}_1 + \frac{\mathbf{u}_0}{\rho_0 c^2} p_1 \right) \right) = 0$$

$$\rho_0 A_0 \left(\frac{\partial \mathbf{u}_1}{\partial t} + \frac{\mathbf{u}_0}{\rho_0 c^2} \frac{\partial p_1}{\partial t} \right) + \nabla \left(A_0 \beta \frac{\mathbf{u}_0^2}{c^2} p_1 + 2 \rho_0 A_0 \beta \mathbf{u}_0 \mathbf{u}_1 \right) \quad (5-4)$$

$$+ A_0 (\nabla p_1 + p_1 \beta_A \nabla p_0) + \tau_w \mathbf{Z} + A \mathbf{F} = 0$$

$$\frac{1}{c^2} = \rho_0 (\beta_0 + \beta_A) = \rho_0 \left(\frac{1}{K_0} + \frac{1}{K_A} \right) = \frac{1}{c_s^2} + \frac{\rho_0}{K_A}$$

where c is the effective speed of sound in the pipe (it includes the effect due to the elastic properties of the pipe defined through K_A). The bulk modulus for the cross-sectional area K_A is given by the pipe material properties according to the so-called Korteweg formula (see Ref. 2). For a system with rigid pipe walls $c_s = c$ as K_A tends to infinity.

Using the fact that the velocity is taken along the tangential direction \mathbf{e}_t , the governing equations are rewritten in terms of the scalar values u and p and projected onto the tangent. The 0 subscript is dropped on the density and area and the 1 subscript is also dropped on the dependent variables.

$$\begin{aligned}
& A \frac{1}{c^2} \frac{\partial p}{\partial t} + \nabla_t \left(A \rho \left(u + \frac{u_0}{\rho c^2} p \right) \right) \cdot \mathbf{e}_t = 0 \\
\rho A \left(\frac{\partial u}{\partial t} + \frac{u_0}{\rho c^2} \frac{\partial p}{\partial t} \right) + \nabla_t \left(A \beta \frac{u_0^2}{c^2} p + 2 \rho A \beta u_0 u \right) \cdot \mathbf{e}_t & \quad (5-5) \\
& + A (\nabla_t p + p \beta_A \nabla_t p_0) \cdot \mathbf{e}_t + \tau_w Z + A (\mathbf{F} \cdot \mathbf{e}_t) = 0 \\
\frac{1}{c^2} = \rho (\beta_0 + \beta_A) = \rho \left(\frac{1}{K_0} + \frac{1}{K_A} \right) = \frac{1}{c_s^2} + \frac{\rho}{K_A}
\end{aligned}$$

where ∇_t is the tangential derivative, τ_w is the tangential wall drag force (SI unit: N/m²) and \mathbf{F} is a volume force (SI unit: N/m³).

GOVERNING EQUATIONS

Pipe Acoustics, Transient Interface

Finally, the expression for the time derivative of the pressure in the momentum equation is replaced by spatial derivatives using the continuity equation. This yields the equations solved in the Pipe Acoustics, Transient interface:

$$\begin{aligned}
& A \frac{1}{c^2} \frac{\partial p}{\partial t} + \nabla_t \left(A \rho \left(u + \frac{u_0}{\rho c^2} p \right) \right) \cdot \mathbf{e}_t = 0 \\
\rho A \frac{\partial u}{\partial t} + \nabla_t \left(A \beta \frac{u_0^2}{c^2} p + 2 \rho A \beta u_0 u \right) \cdot \mathbf{e}_t + u_0 \nabla_t \left(A \rho \left(u + \frac{u_0}{\rho c^2} p \right) \right) \cdot \mathbf{e}_t & \quad (5-6) \\
& + A (\nabla_t p + p \beta_A \nabla_t p_0) \cdot \mathbf{e}_t + \tau_w Z + A (\mathbf{F} \cdot \mathbf{e}_t) = 0 \\
\frac{1}{c^2} = \rho (\beta_0 + \beta_A) = \rho \left(\frac{1}{K_0} + \frac{1}{K_A} \right) = \frac{1}{c_s^2} + \frac{\rho}{K_A}
\end{aligned}$$

Pipe Acoustics, Frequency Domain Interface

In the frequency domain all variables are assumed to be time harmonic such that

$$\begin{aligned}
p &= \tilde{p}(\mathbf{x}) e^{i\omega t} \\
u &= \tilde{u}(\mathbf{x}) e^{i\omega t}
\end{aligned} \quad (5-7)$$

inserting this into the governing [Equation 5-6](#) (and dropping the tilde) yields the equations solved in the Pipe Acoustics, Frequency Domain interface:

$$\begin{aligned}
& i\omega \frac{A}{c^2} p + \nabla_t \left(A \rho \left(u + \frac{u_0}{\rho c^2} p \right) \right) \cdot \mathbf{e}_t = 0 \\
& i\omega \rho A u + \nabla_t \left(A \beta \frac{u_0^2}{c^2} p + 2\rho A \beta u_0 u \right) \cdot \mathbf{e}_t + u_0 \nabla_t \left(A \rho \left(u + \frac{u_0}{\rho c^2} p \right) \right) \cdot \mathbf{e}_t \\
& \quad + A (\nabla_t p + p \beta_A \nabla_t p_0) \cdot \mathbf{e}_t + \tau_w \mathbf{Z} + A (\mathbf{F} \cdot \mathbf{e}_t) = 0 \\
& \frac{1}{c^2} = \rho (\beta_0 + \beta_A) = \rho \left(\frac{1}{K_0} + \frac{1}{K_A} \right) = \frac{1}{c_s^2} + \frac{\rho}{K_A}
\end{aligned} \tag{5-8}$$

where $\omega = 2\pi f$ is the angular frequency and f is the frequency.

Theory for the Pipe Acoustics Boundary Conditions

PRESSURE, OPEN, AND CLOSED CONDITIONS

The simplest boundary conditions to specify are to prescribe the pressure or the velocity at the pipe ends. These result in the [Pressure](#) condition

$$p = p_{\text{in}}$$

and the [Velocity](#) condition

$$u = u_{\text{in}}$$

and can be set independently of each other leaving the other dependent variable free.

A special subclass of the velocity condition is the [Closed](#) condition, where

$$u = 0$$

corresponds to the sound-hard wall condition in pressure acoustics. It is also assumed here that $u_0 = 0$ at a closed boundary.

END IMPEDANCE CONDITION

At the end of pipes the relation between the pressure and the velocity can be defined in terms of an end impedance Z_{end} . The [End Impedance](#) condition is in the Pipe Acoustics interface given by

$$A \left(\rho u + \frac{u_0}{c^2} p \right) = A \left(\rho \frac{1}{Z_{\text{end}}} + \frac{u_0}{c^2} \right) p \tag{5-9}$$

where $Z_{\text{end}} = p/u$ (SI unit: (Pa·s)/m). Different models for the end impedance exist in the Pipe Acoustics interfaces. The variety depend on if the transient or the frequency domain equations are solved.

Transient End-Impedance Models

In the transient version of the physics interface the end impedance can be user-defined or set to mimic an infinite long pipe for low Mach number background flow conditions. In this case it is assumed that the pipe continues with constant cross section A and that there is no external body force \mathbf{F} and drag τ_w . Because the acoustic waves are, by design, always normal to the pipe ends. In order to define the relation between the pressure and the velocity (the impedance) the dispersion relation for a plane wave needs to be determined.

In order to do so insert the assumed plane waveform

$$\begin{aligned} p &= \text{Re}(\tilde{p}e^{i(\omega t - kx)}) \\ u &= \text{Re}(\tilde{u}e^{i(\omega t - kx)}) \end{aligned}$$

into the governing [Equation 5-6](#) and solve for the desired relations. After some manipulation this results in

$$\frac{1}{Z_{\text{end}}} = \frac{u}{p} = \frac{1}{c^2 \rho} \left(\frac{\omega}{k} - u_0 \right)$$

with the dispersion relation

$$\frac{\omega}{k} = \beta u_0 \pm c \sqrt{\beta(\beta - 1) \left(\frac{u_0}{c} \right)^2 + \left(1 - \frac{1}{\beta} \beta_A \nabla p_0 \right)} \quad (5-10)$$

This dispersion relation is nonlinear in k . In the limit where β_A tends to zero and for small Mach numbers $M (= u_0/c)$, the expression is expanded to

$$\frac{\omega}{k} \cong \beta u_0 \pm c \left(1 + \frac{1}{2} \beta(\beta - 1) \left(\frac{u_0}{c} \right)^2 \right)$$

Hence, the infinite pipe (low Mach number limit) end impedance relation reads

$$\frac{1}{Z_{\text{end}}} = \frac{1}{c^2 \rho} \left[(\beta - 1) u_0 \pm c \left(1 + \frac{1}{2} \beta(\beta - 1) \left(\frac{u_0}{c} \right)^2 \right) \right] \quad (5-11)$$

where the sign in front of c depends on the direction of propagation of the wave.

Frequency Domain End-Impedance Models

In the frequency domain many engineering relations exist for the end impedance or radiation impedance of a pipe or waveguide. Most of the relations apply only to a specific geometry or frequency range. The relations available in the Pipe Acoustics, Frequency Domain interface are:

- *Infinite pipe (low Mach number limit)*: This is the same relation as for the transient study and the end impedance is given by Equation 5-11. This can be thought of as the characteristic impedance of the tube.
- *Infinite pipe*: This relation uses the full dispersion relation given in Equation 5-10 and yields the expression

$$\frac{1}{Z_{\text{end}}} = \frac{1}{c^2 \rho} \left[(\beta - 1)u_0 \pm c \sqrt{\beta(\beta - 1) \left(\frac{u_0}{c}\right)^2 + \left(1 - \frac{1}{k} \beta_A \nabla p_0\right)} \right] \quad (5-12)$$

where the wave number k at the right hand side is a user input. In the frequency domain a good estimate for this quantity is simply ω/c .

- *Flanged pipe, circular*: In the case of a circular pipe terminated in an infinite baffle (a flanged pipe) an analytical expression exists for the radiation impedance (see Ref. 1),

$$Z_{\text{end}} = \rho c \left(1 - \frac{2J_1(2ka)}{2ka} + i \frac{2H_1(2ka)}{2ka} \right) \quad (5-13)$$

where J_1 is the Bessel function of order 1, H_1 is the Struve function of order 1, a is the pipe radius, and k is the wave number. The Struve function is approximated according to Ref. 3 by

$$H_1(x) \approx \frac{2}{\pi} - J_0(x) + \left(\frac{16}{\pi} - 5\right) \frac{\sin x}{x} + \left(12 - \frac{36}{\pi}\right) \frac{1 - \cos x}{x^2} \quad (5-14)$$

In the low frequency limit (small ka) Equation 5-13 reduces to the classical expression for the radiation impedance

$$Z_{\text{end}} = \rho c \left(\frac{1}{2} (ka)^2 + i(0,8216 \cdot ka) \right) \quad (5-15)$$

- *Flanged pipe, rectangular*: In the case of a pipe of rectangular cross-section (with sides w_1 and h_1) terminated in an infinite baffle (a flanged pipe) the radiation impedance can be approximated by

$$Z_{\text{end}} = \frac{\rho c}{2\pi} \left(k^2 (w_i h_i)^2 + i k (w_i h_i)^{3/2} f\left(\frac{w_i}{h_i}\right) \right) \quad k w_i \ll 1, \quad k h_i \ll 1 \quad (5-16)$$

$$f(x) = 2x^{1/2} \sinh^{-1} x^{-1} + 2x^{-1/2} \sinh^{-1} x + \frac{2}{3} x^{3/2} + \frac{2}{3} x^{-3/2} - \frac{2}{3} (x + x^{-1})^{3/2}$$

see [Ref. 4](#) and [Ref. 5](#).

- *Unflanged pipe, circular (low ka limit)*: In the case of a circular pipe of radius a ending in free air the classical low ka limit for the radiation impedance is given by

$$Z_{\text{end}} = \rho c \left(\frac{1}{4} (ka)^2 + i(0,6133 \cdot ka) \right) \quad ka \ll 1 \quad (5-17)$$

see [Ref. 1](#) and [Ref. 5](#).

- *Unflanged pipe, circular*: A solution for the unflanged pipe exists for the case when $ka \ll 3,83 = 1,22\pi$, it is presented in [Ref. 6](#) and is based on solving the Wiener-Hopf integral, it reads

$$Z_{\text{end}} = \rho c \frac{1+R}{1-R} \quad R = |R| e^{2ika\delta}$$

$$|R| = e^{-(ka)^2/2} \left(1 + \frac{1}{6} (ka)^4 \left[\ln \left(\frac{1}{\gamma ka} + \frac{19}{12} \right) \right] \right) \quad \gamma = e^{0,5772} \quad ka < 1 \quad (5-18)$$

$$|R| = \sqrt{\pi ka} e^{-ka} \left(1 + \frac{3}{32} \frac{1}{(ka)^2} \right) \quad 1 < ka < 3,83$$

where δ is an interpolation function found by numerical integration for $ka = 0$, $\delta = 0.6133$.

Common for the last four radiation impedance relations is that they do only apply when there is no background flow present $u_0 = 0$ (or at least when it is very small).

Solving Transient Problems

When solving transient acoustic problems where the wave shape is not necessarily harmonic it might be necessary to resolve its spatial variations with a fine mesh, say with a minimal scale dx . Now, in order for the numerical solution of the temporal development of the acoustic field to be good it is necessary to restrict the maximal time steps dt taken by the solver. The condition is known as the CFL condition (Courant–Friedrichs–Lewy condition). For transient acoustic problems it is defined as

$$C = c \cdot \frac{dt}{dx}$$

where C is the Courant number, and c is the velocity.

For applications where all the shape functions are quadratic the Courant number should be around 0.2. This condition restricts any acoustic disturbances to propagate more than 20% of the mesh size dx during one time step dt . In the Pipe Acoustics interface where a mixed formulation exists, with linear elements for the pressure and quadratic elements for the velocity, the condition might have to be tightened such that $C < 0.2$.



For an example where the CFL condition is used see *Water Hammer*: Application Library path **Pipe_Flow_Module/Verification_Examples/water_hammer_verification**.

Cut-off Frequency

The Pipe Acoustics interface assumes plane wave propagation. This means that it cannot model the propagation of the higher order modes that can propagate above their cut-off frequency f^c . In a rectangular pipe of cross section width w_1 and height h_1 the cut-off frequency is

$$f_{mn}^c = \frac{1}{2}c \sqrt{\left(\frac{m}{w_1}\right)^2 + \left(\frac{n}{h_1}\right)^2}$$

In a pipe of circular cross section (with radius a) the cut-off frequency is

$$f_{mn}^c = \frac{\alpha'_{mn}c}{2\pi a}$$

where α'_{mn} is the n 'th zero of the differential of the Bessel function $J'_m(x)$ or order m . The first few values are $\alpha'_{01} = 0$, $\alpha'_{02} = 3.83$, $\alpha'_{11} = 1.84$, and $\alpha'_{21} = 3.05$ (see [Ref. 1](#) and [Ref. 5](#) for further details).

Flow Profile Correction Factor β

The flow profile correction factor β accounts for the ratio of the integrated local square velocity field to the square of the integrated local velocity field (see [Equation 5-3](#)). It is defined in terms of the total velocity field (background plus acoustic variations).

In the case of no-background flow ($u_0 = 0$) β is 1 in the absence of a wall drag coefficient, as only plane wave modes propagate. If a wall drag force is introduced, to

model some loss mechanism, β starts to differ slightly from 1. This can for example be losses introduced to model viscous and thermal effects in narrow pipes.

In the presence of a background u_0 the factor β can be set different 1 in order to model a non-flat velocity profile inside the tube. The value of b (and the actual shape of the background field) influences the convective momentum transfer balances. The places where β enter the governing equations are multiplied with either the Mach number or the Mach number squared, indicating that the effects become important for an increasing background flow.

References for the Pipe Acoustics Interfaces

1. D.T. Blackstock, *Fundamentals of Physical Acoustics*, John Wiley & Sons, 2000.
2. M.S. Ghidaoui, M. Zhao, D.A. McInnis, and D.H. Axworthy, “A Review of Water Hammer Theory and Practice”, *Applied Mechanics Reviews*, ASME, 2005.
3. R.M. Aarts and A.J.E.M. Janssen, “Approximation of the Struve Function H_1 Occurring in Impedance Calculations”, *J. Acoust. Soc. Am.*, vol. 113, pp. 2635–2637, 2003.
4. O.A. Lindemann, “Radiation Impedance of a Rectangular Piston at Very Low Frequencies”, *J. Acoust. Soc. Am.*, vol 44, pp. 1738–1739, 1968.
5. A.D. Pierce, *Acoustics: An Introduction to its Physical Principles and Applications*, Acoustics Society of America, 1994.
6. H. Levine and J. Schwinger, “On the Radiation of Sound from an Unflanged Circular Pipe”, *Phys. Rev.*, vol. 73, pp. 383–406, 1948.

Index

- A** Application Libraries window 13
 - application library examples
 - nonisothermal pipe flow 40
 - pipe acoustics, transient 116
 - pipe flow 20
 - water hammer 44
- B** bend (node) 28
 - bend specification 28
 - Bingham model 55
 - boundary nodes
 - heat transfer in pipes 66
 - nonisothermal pipe flow 40
 - pipe acoustics 101
 - pipe flow 20
 - reacting pipe flow 88
 - transport of diluted species in pipes 83
 - water hammer 44
 - bulk modulus 110
- C** CFL condition 115
 - closed (node)
 - pipe acoustics 104
 - water hammer 46
 - Colebrook's explicit formula 51
 - common settings 11
 - compressible flow 41
 - concentration (node) 85
 - contraction/expansion (node) 29
 - convection and diffusion (node)
 - reacting pipe flow 91
 - transport of diluted species in pipes 84
 - Courant number 116
- D** Darcy friction factor 47
 - expressions for 50
 - dimensionless numbers
 - Grashof 80
 - Hedström 55
 - modified Reynolds 54
 - Nusselt 78
 - Prandtl 79
 - Rayleigh 80
 - Reynolds 49
- documentation 12
- E** edge nodes
 - heat transfer in pipes 66
 - non-isothermal pipe flow 40
 - pipe acoustics 101
 - pipe flow 20
 - reacting pipe flow 88
 - transport of diluted species in pipes 83
 - water hammer 44
- emailing COMSOL 14
- end impedance (node)
 - pipe acoustics 105
- extended Einstein 53
- external film resistance (node) 70
- external film resistance, theory 79
- F** flow resistance 24, 49
 - fluid (node)
 - nonisothermal pipe flow 41
 - reacting pipe flow 90
 - fluid models 19
 - fluid properties (node) 21, 91
 - pipe acoustics 102
 - forced convection 79
- G** gas-liquid, effective Reynolds number 53
- gas-liquid, friction factor multiplier 52
- Gnielinski equation for Nusselt number 79
- Grashof number 80
- H** Haaland equation 51

- heat outflow (node) 72
- heat source (node) 67
- heat transfer (node)
 - heat transfer in pipes 67
 - nonisothermal pipe flow 42
- heat transfer in pipes interface 65
 - theory 73
- Hedström number 55
- I** initial values (node)
 - heat transfer in pipes 72
 - pipe acoustics, frequency domain 102
 - pipe acoustics, transient 102
 - pipe flow 25
 - reacting pipe flow 90
 - transport of diluted species in pipes 84
- inlet (node) 26
- internal film resistance (node) 69
- internal film resistance, theory 78
- internet resources 11
- K** knowledge base, COMSOL 14
- Korteweg formula 61, 110
- L** local friction loss (node) 46
 - loss coefficients 49
 - lossless fitting (node) 28
- M** mass outflow (node) 86
 - momentum and continuity equations 47
 - MPH-files 13
- N** no flow (node) 27
 - nodes, common settings 11
 - nonisothermal pipe flow interface 39
 - theory 47
 - Nusselt number 78
 - n-way junction (node) 36
 - n-way junctions 49
- O** outlet (node) 27
- P** pair nodes
 - heat transfer in pipes 66
 - nonisothermal pipe flow 40
 - pipe acoustics 101
 - pipe flow 20
 - reacting pipe flow 88
 - transport of diluted species in pipes 83
 - water hammer 44
- phase fractions (node) 25
- physics interfaces, common settings 11
- physics symbols
 - showing 37
- pipe acoustics, frequency domain (pafd)
 - interface 98
- pipe acoustics, transient (patd) interface
 - 100
 - theory 108
- pipe connection (node) 37
- pipe cross section area 73
- pipe flow interface 19
 - theory 47
- pipe properties (node)
 - pipe acoustics 103
 - pipe flow 23
 - reacting pipe flow 89
 - water hammer 45
- point nodes
 - heat transfer in pipes 66
 - nonisothermal pipe flow 40
 - pipe acoustics 101
 - pipe flow 20
 - reacting pipe flow 88
 - transport of diluted species in pipes 83
 - water hammer 44
- Prandtl number 79
- pressure (node)
 - pipe acoustics 104
 - pipe flow 28
 - water hammer 46

- Pressure Work 41
- pump (node) 30
- R** Rayleigh number 80
- reacting pipe flow interface 87
 - theory 73, 93
- reactions (node) 85
- Reynolds number 49
- Reynolds number, modified 54
- S** shell balance 75
- showing
 - physics symbols 37
- single-phase fluids 50
- stabilization 80
- standard settings 11
- Stokes formula 51
- surface roughness 24
 - values for 56
- Swamee-Aggarwal equation 55
- Swamee-Jain equation 52
- swirl-correction factor 109
- symbols for physics 37
- T** T- junction (node) 31
- tangential velocity 48
- Taylor series 109
- technical support, COMSOL 14
- temperature (node) 72
- theory
 - heat transfer in pipes 73
 - non-isothermal pipe flow 47
 - pipe acoustics, transient interface 108
 - pipe flow 47
 - reacting pipe flow 73, 93
 - transport of diluted species in pipes 93
 - water hammer 60
- T-junctions 49
- transport of diluted species in pipes in-
terface 82
 - theory 93
- two-phase flow 52
- two-phase friction factor 53
- U** unit tangent vector 48
- V** valve (node) 29
- velocity (node)
 - pipe acoustics 105
 - water hammer 46
- volume force (node) 23
- volumetric void fraction 53
- von Karman formula 51
- W** wall heat transfer (node) 68
- wall heat transfer, theory 74
- wall layer (node) 71
- wall mass transfer (node) 85
- water hammer interface 43
 - theory 60
- websites, COMSOL 14
- Wood equation 51
- Y** Y- junction 34
- Y-junctions 49

