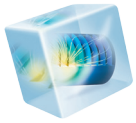


Pipe Flow Module

User's Guide

VERSION 4.3

COMSOL

Pipe Flow Module User's Guide

Version: May 2012 COMSOL 4.3

Contact Information

Visit www.comsol.com/contact for a searchable list of all COMSOL offices and local representatives. From this web page, search the contacts and find a local sales representative, go to other COMSOL websites, request information and pricing, submit technical support queries, subscribe to the monthly eNews email newsletter, and much more.

If you need to contact Technical Support, an online request form is located at www.comsol.com/support/contact.

Other useful links include:

- Technical Support www.comsol.com/support
- Software updates: www.comsol.com/support/updates
- Online community: www.comsol.com/community
- Events, conferences, and training: www.comsol.com/events
- Tutorials: www.comsol.com/products/tutorials
- Knowledge Base: www.comsol.com/support/knowledgebase

Part No. CM022801

Contents

Chapter 1: Introduction

Chapter 2: The Fluid Flow Branch

Chapter 3: The Heat Transfer Branch

Chapter 4: The Chemical Species Transport Branch

Chapter 5: The Acoustics Branch

Chapter 6: Materials

1

Introduction

This guide describes the Pipe Flow Module, an optional add-on package for COMSOL Multiphysics designed to model and simulate incompressible flow, heat and mass transfer in pipes and channels with the Pipe Flow physics. Compressible hydraulic transients and acoustics waves can also be modeled using the Water Hammer physics and Pipe Acoustics physics, 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 pipelines 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.

In this chapter:

About the Pipe Flow Module

These topics are included in this section:

What Can the Pipe Flow Module Do?

The Pipe Flow Module is intended for the modeling and simulation of flow of incompressible 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 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 pipelines in the oil and gas industry.

In common for pipes and channels that can be modeled 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 module includes these physics:

- The *Pipe Flow* physics computes the pressure and velocity field in isothermal pipe systems.
- The *Heat Transfer in Pipes* physics 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* physics 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 *Non-Isothermal Pipe Flow* physics is a multiphysics interface that solves the flow, pressure, and temperature simultaneously and fully coupled.

- The *Reacting Pipe Flow* physics is a multiphysics interface that solves the flow, pressure, temperature, and reacting species transport simultaneously and fully coupled.
- The *Water Hammer* physics solves rapid hydraulic transients in pipe systems, taking the elastic properties of both the fluid and pipe wall into account.
- The *Pipe Acoustics, Transient* physics models sound waves in flexible pipe systems.

The physics in the module define the conservation of momentum, energy, and mass of an 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 are used that 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.

Pipe Flow Module Interfaces

INTERFACE	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Acoustics				
Acoustic-Structure Interaction				
Pipe Acoustics, Transient Requires both the Pipe Flow Module and Acoustics Module.		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

INTERFACE	ICON	TAG	SPACE DIMENSION	PRESET STUDIES
Fluid Flow				
Single-Phase Flow				
Pipe Flow		pfl	3D, 2D	stationary; time dependent
Reacting Pipe Flow		rpfl	3D, 2D	stationary; time dependent
Water Hammer		whtd	3D, 2D	time dependent
Non-Isothermal Pipe Flow		nipfl	3D, 2D	stationary; time dependent
Heat Transfer				
Heat Transfer in Pipes		htp	3D, 2D	stationary; time dependent

Show More Physics Options

There are several features available on many physics interfaces or individual nodes. This section is a short overview of the options and includes links to the *COMSOL Multiphysics User's Guide* or *COMSOL Multiphysics Reference Guide* where additional information is available.

Important	The links to the features described in the *COMSOL Multiphysics User's Guide* and *COMSOL Multiphysics Reference Guide* do not work in the PDF, only from within the online help.

Tip	To locate and search all the documentation for this information, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

To display additional features for the physics interfaces and feature nodes, click the **Show** button ( ) on the **Model Builder** and then select the applicable option.

After clicking the **Show** button ( ), some sections display on the settings window when a node is clicked and other features are available from the context menu when a node is right-clicked. For each, the additional sections that can be displayed include **Equation**, **Advanced Settings**, **Discretization**, **Consistent Stabilization**, and **Inconsistent Stabilization**.

You can also click the **Expand Sections** button ( ) in the **Model Builder** to always show some sections or click the **Show** button ( ) and select **Reset to Default** to reset to display only the **Equation** and **Override and Contribution** sections.

For most physics nodes, both the **Equation** and **Override and Contribution** sections are always available. Click the **Show** button ( ) and then select **Equation View** to display the **Equation View** node under all physics nodes in the **Model Builder**.

Availability of each feature, and whether it is described for a particular physics node, is based on the individual physics selected. For example, the **Discretization**, **Advanced Settings**, **Consistent Stabilization**, and **Inconsistent Stabilization** sections are often described individually throughout the documentation as there are unique settings.

SECTION	CROSS REFERENCE	LOCATION IN COMSOL MULTIPHYSICS USER'S GUIDE OR REFERENCE GUIDE
Show More Options and Expand Sections	• Showing and Expanding Advanced Physics Sections • The Model Builder Window	User's Guide
Discretization	• Show Discretization • Element Types and Discretization	User's Guide
	• Finite Elements • Discretization of the Equations	Reference Guide
Discretization - Splitting of complex variables	Compile Equations	Reference Guide
Pair Selection	• Identity and Contact Pairs • Specifying Boundary Conditions for Identity Pairs	User's Guide
Consistent and Inconsistent Stabilization	Show Stabilization	User's Guide
	• Stabilization Techniques • Numerical Stabilization	Reference Guide

SECTION	CROSS REFERENCE	LOCATION IN COMSOL MULTIPHYSICS USER'S GUIDE OR REFERENCE GUIDE
Geometry	Working with Geometry	User's Guide
Constraint Settings	Using Weak Constraints	User's Guide

Where Do I Access the Documentation and Model Library?

A number of Internet resources provide more information about COMSOL Multiphysics, including licensing and technical information. The electronic documentation, Dynamic Help, and the Model Library are all accessed through the COMSOL Desktop.

Important

If you are reading the documentation as a PDF file on your computer, the blue links do not work to open a model or content referenced in a different user's guide. However, if you are using the online help in COMSOL Multiphysics, these links work to other modules, model examples, and documentation sets.

THE DOCUMENTATION

The *COMSOL Multiphysics User's Guide* and *COMSOL Multiphysics Reference Guide* describe all interfaces and functionality included with the basic COMSOL Multiphysics license. These guides also have instructions about how to use COMSOL Multiphysics and how to access the documentation electronically through the COMSOL Multiphysics help desk.

To locate and search all the documentation, in COMSOL Multiphysics:

- Press F1 for Dynamic Help,
- Click the buttons on the toolbar, or
- Select **Help>Documentation** () or **Help>Dynamic Help** () from the main menu

and then either enter a search term or look under a specific module in the documentation tree.

THE MODEL LIBRARY

Each model comes with documentation that includes a theoretical background and step-by-step instructions to create the model. The models are available in COMSOL

as MPH-files that you can open for further investigation. You can use the step-by-step instructions and the actual models as a template for your own modeling and applications.

SI units are used to describe the relevant properties, parameters, and dimensions in most examples, but other unit systems are available.

To open the Model Library, select **View>Model Library** () from the main menu, and then search by model name or browse under a module folder name. Click to highlight any model of interest, and select **Open Model and PDF** to open both the model and the documentation explaining how to build the model. Alternatively, click the **Dynamic Help** button () or select **Help>Documentation** in COMSOL to search by name or browse by module.

The model libraries are updated on a regular basis by COMSOL in order to add new models and to improve existing models. Choose **View>Model Library Update** () to update your model library to include the latest versions of the model examples.

If you have any feedback or suggestions for additional models for the library (including those developed by you), feel free to contact us at info@comsol.com.

CONTACTING COMSOL BY EMAIL

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

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 case number is sent to you by email.

COMSOL WEB SITES

Main Corporate web site	www.comsol.com
Worldwide contact information	www.comsol.com/contact
Technical Support main page	www.comsol.com/support
Support Knowledge Base	www.comsol.com/support/knowledgebase
Product updates	www.comsol.com/support/updates
COMSOL User Community	www.comsol.com/community

Typographical Conventions

All COMSOL user's guides use a set of consistent typographical conventions that make it easier to follow the discussion, understand what you can expect to see on the graphical user interface (GUI), and know which data must be entered into various data-entry fields.

In particular, these conventions are used throughout the documentation:

CONVENTION	EXAMPLE
text highlighted in blue	Click text highlighted in blue to go to other information in the PDF. When you are using the online help desk in COMSOL Multiphysics, these links also work to other modules, model examples, and documentation sets.
boldface font	A **boldface** font indicates that the given word(s) appear exactly that way on the COMSOL Desktop (or, for toolbar buttons, in the corresponding tip). For example, the **Model Builder** window () is often referred to and this is the window that contains the model tree. As another example, the instructions might say to click the **Zoom Extents** button (), and this means that when you hover over the button with your mouse, the same label displays on the COMSOL Desktop.
Forward arrow symbol **>**	The forward arrow symbol **>** is instructing you to select a series of menu items in a specific order. For example, **Options>Preferences** is equivalent to: From the **Options** menu, choose **Preferences**.
`Code` (monospace) font	A `Code` (monospace) font indicates you are to make a keyboard entry in the user interface. You might see an instruction such as "Enter (or type) `1.25` in the **Current density** field." The monospace font also is an indication of programming code or a variable name.

CONVENTION	EXAMPLE
Italic *Code* (monospace) font	An italic *Code* (monospace) font indicates user inputs and parts of names that can vary or be defined by the user.
Arrow brackets <> following the Code (monospace) or *Code* (italic) fonts	The arrow brackets included in round brackets after either a monospace Code or an italic *Code* font means that the content in the string can be freely chosen or entered by the user, such as feature tags. For example, `model.geom(<tag>)` where <*tag*> is the geometry's tag (an identifier of your choice). When the string is predefined by COMSOL, no bracket is used and this indicates that this is a finite set, such as a feature name.

KEY TO THE GRAPHICS

Throughout the documentation, additional icons are used to help navigate the information. These categories are used to draw your attention to the information based on the level of importance, although it is always recommended that you read these text boxes.

ICON	NAME	DESCRIPTION
	Caution	A Caution icon is used to indicate that the user should proceed carefully and consider the next steps. It might mean that an action is required, or if the instructions are not followed, that there will be problems with the model solution.
	Important	An Important icon is used to indicate that the information provided is key to the model building, design, or solution. The information is of higher importance than a note or tip, and the user should endeavor to follow the instructions.
	Note	A Note icon is used to indicate that the information may be of use to the user. It is recommended that the user read the text.
	Tip	A Tip icon is used to provide information, reminders, shortcuts, suggestions of how to improve model design, and other information that may or may not be useful to the user.
	See Also	The See Also icon indicates that other useful information is located in the named section. If you are working on line, click the hyperlink to go to the information directly. When the link is outside of the current PDF document, the text indicates this, for example See The Laminar Flow Interface in the *COMSOL Multiphysics User's Guide*. Note that if you are in COMSOL Multiphysics' online help, the link will work.

ICON	NAME	DESCRIPTION
	Model	The Model icon is used in the documentation as well as in COMSOL Multiphysics from the View>Model Library menu. If you are working online, click the link to go to the PDF version of the step-by-step instructions. In some cases, a model is only available if you have a license for a specific module. These examples occur in the COMSOL Multiphysics User's Guide. The Model Library path describes how to find the actual model in COMSOL Multiphysics, for example If you have the RF Module, see Radar Cross Section: Model Library path **RF_Module/Tutorial_Models/radar_cross_section**
Space Dimension		Another set of icons are also used in the Model Builder—the model space dimension is indicated by 0D , 1D , 1D axial symmetry  , 2D  , 2D axial symmetry  , and 3D  icons. These icons are also used in the documentation to clearly list the differences to an interface, feature node, or theory section, which are based on space dimension.

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 User's Guide*.


Tip

As detailed in the section Where Do I Access the Documentation and Model Library? this information is also searchable 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 ACOUSTICS BRANCH

The Acoustics Branch chapter describes the Pipe Acoustics, Transient interface and its theory. This interface requires both the Acoustics Module and the Pipe Flow Module.

THE FLUID FLOW BRANCH

The The Fluid Flow Branch chapter describes the Pipe Flow interface and the Non-Isothermal Pipe Flow interface including the underlying theory for these interfaces.

THE HEAT TRANSFER BRANCH

The The Heat Transfer Branch chapter describes the Heat Transfer in Pipes interface and the underlying theory for the interface.

2

The Fluid Flow Branch

This chapter has information about the interfaces in the Pipe Flow Module available under the **Fluid Flow** branch ( ) in the **Model Wizard**.

In this chapter:

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 ( ) in the **Model Wizard**, is used to model incompressible fluid flow in piping systems. The equations that are solved are 1D equations that live on 2D boundary segments or 3D edge segments. The **Non-Isothermal Pipe Flow** interface ( ), found under the **Non-Isothermal Flow** branch ( ) in the **Model Wizard**, solves a temperature equation together with the equations for incompressible flow. The **Water Hammer** interface ( ), found under the **Fluid Flow>Single-Phase Flow** branch ( ) in the **Model Wizard**, 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** interface ( ), found under the **Single-Phase Flow** branch ( ) in the **Model Wizard**, has the equations, boundary conditions, and volume forces for modeling incompressible fluid flow in one dimension.

When this interface is added, these default nodes are also added to the **Model Builder**—**Fluid Properties**, **Pipe Properties**, **Pressure**, and **Initial Values**. Right-click the **Pipe Flow** node to add other features that implement, for example, boundary conditions and volume forces.

Model	• Discharging Tank: Model Library path **Pipe_Flow_Module/Tutorial_Models/discharging_tank** • Convective Flow in a Heat Exchanger Plate: Model Library path **Pipe_Flow_Module/Heat_Transfer/heat_exchanger_plate**

2D *3D*	The interface is available in 3D on edges and points, and in 2D on boundaries and points.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `pfl`.

EDGE OR BOUNDARY SELECTION

The default setting is to include **All edges** or **All boundaries** in the model. To choose specific edges or boundaries, select **Manual** from the **Selection** list.

DEPENDENT VARIABLES

This 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)

For an explanation of the tangential velocity, see further in the section Theory for the Pipe Flow Interface.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. It controls the discretization (the element types used in the finite element formulation).

Select discretization options from the **Pressure** and **Tangential velocity** lists—**Quadratic** (the default), **Linear**, **Cubic**, or **Quartic**.

For each **Dependent variable** in the table under **Value types when using splitting of complex variables**, choose either a **Complex** or **Real Value type**. Click the cell to select from a drop-down list.

See Also

- Show More Physics Options
- Edge, Boundary, Point, and Pair Features for the Pipe Flow Interface
- Theory for the Pipe Flow Interface

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

2D

3D

The interface is available in 3D on edges and points, and in 2D on boundaries and points.

The Pipe Flow Interface has these boundary, edge, point, and pair features available and listed in alphabetical order:

- Bend
- Contraction / Expansion

- Initial Values
- Inlet
- No Flow
- Outlet
- Pipe Properties
- Pressure
- Pump
- T-Junction
- Valve

These features are described for the **Laminar Flow** interface in the *COMSOL Multiphysics User's Guide*:

- Fluid Properties
- Volume Force


Note

For this interface, apply the **Fluid Properties** and **Volume Force** node to **Edges** (3D models) and **Boundaries** (2D models) instead of **Domains**.


See Also

In the *COMSOL Multiphysics User's Guide*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs

Pipe Properties

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

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the pipe properties.

PIPE SHAPE

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

- If **Round** is selected, enter a value or expression for the **Inner diameter** d_i (SI unit: m). The default is 10 cm.
- If **Square** is selected, enter a value or expression for the **Inner width** w_i (SI unit: m). The default is 5 cm.
- If **Rectangular** is selected, 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 10cm).
- If **User defined** is selected, enter a value or expression for the **Cross sectional area** A_c (SI unit: m^2. The default is 0.01 m^2) and **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**.

- If **Churchill**, **Wood**, **Haaland**, **Von Karman**, or **Swamee-Jain** is selected, go to the Surface Roughness section to select from a list of predefined values. Alternatively, enter values or expressions.
- If **User defined** is selected, enter a value or expression for the **Darcy friction factor** f_D (unitless).

Non-Newtonian fluids

For Power-law and Bingham fluids in round tubes, the **Irvine**, **Stokes**, and **Darby** friction models are available.

For more information about the friction models, please refer to the theory section Expressions for the Darcy Friction Factor.

SURFACE ROUGHNESS


Note

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—**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**.

- If **User defined** is selected, enter a value or expression for the **Roughness** (SI unit: m).

Initial Values

The **Initial Values** feature 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.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define initial values.

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.


Tip

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.

POINT SELECTION

From the **Selection** list, choose the points that represent inlets.

INLET SPECIFICATION

Select a **Specification** for the inlet—**Mass flow rate** (the default), **Velocity**, or **Volumetric flow rate**.

- If **Mass flow rate** is selected, enter a value or expression for the **Mass flow rate** $q_{m,0}$ (SI unit: kg/s). The default is 0.
- If **Velocity** is selected, enter a value or expression for the **Velocity** u_0 (SI unit: m/s). The default is 0.
- If **Volumetric flow rate** is selected, enter a value or expression for the **Volumetric flow rate** $q_{v,0}$ (SI unit: m^3/s). The default is 0.

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.


Note

See Inlet for all settings.


Tip

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).

POINT SELECTION

From the **Selection** list, choose the points that represent no flow.

Pressure

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


Note

The pressure condition can only be applied to a point which is connected to exactly one edge (inlet or outlet of a system).

POINT SELECTION


Note

For the default node no user selection is required. **All points** is automatically selected.

When additional nodes are added, from the **Selection** list choose the points that represent the pressure.

BOUNDARY PRESSURE

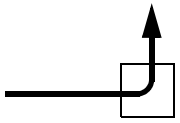
Enter a value or expression for the **Pressure** p_0 (SI unit: Pa). The default is 101325 Pa.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Bend

Use the **Bend** node to introduce a pressure drop in a point associated with a bend in the pipe system.



Note

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

POINT SELECTION

From the **Selection** list, choose the points that represent the bend.

BEND SPECIFICATION

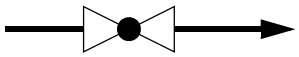
Select a **Bend**—**90 degrees standard elbow (K = 0.9)** (the default), **45 degrees standard elbow (K = 0.5)**, or **User specified loss coefficient**. If **User specified loss coefficient** is selected, also enter a **Loss coefficient** K_f (unitless).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. If required, select the **Use weak constraints** check box.

Valve

Use the **Valve** node to introduce a pressure drop 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.



Note

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

POINT SELECTION

From the **Selection** list, choose the points for the valve.

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 specified loss coefficient**. For all selections, enter values or expressions for the **First edge** i and **Second edge** j (unitless numbers). If **User specified loss coefficient** is selected, also enter a **Loss coefficient** K_f (unitless).

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. If required, select the **Use weak constraints** check box.

Contraction / Expansion

Use the **Contraction/Expansion** node to introduce a pressure drop in a point associated with the position of a contraction or expansion in the pipe system.

Note

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

POINT SELECTION

From the **Selection** list, choose the points for the contraction.

CONTRACTION SPECIFICATION

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

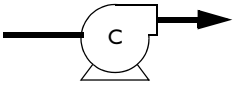
- If **Gradual** is selected, also enter an **Angle of convergence** α (SI unit: radians). The default is 0.
- If **User specified loss coefficient** is selected, also enter a **Loss coefficient** K_f (unitless). The default is 0.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. If required, select the **Use weak constraints** check box.

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.



Note

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

POINT SELECTION

From the **Selection** list, choose the points for the pump.

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**.

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

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

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. If required, select the **Use weak constraints** check box.

T-Junction

Use the **T-Junction** node to specify the friction coefficients for a T-junction, which can act as a split or a merger. The pressure drop is calculated according to Equation 2-9.

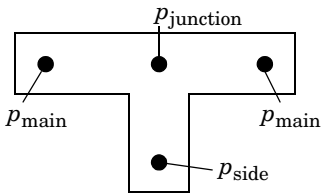


Figure 2-1: A T-junction with local pressure indications.

Note

The **T-junction** can only be applied to a point that is connected to exactly two collinear main branches and one perpendicular side branch. The flow can go in either direction through each of the main branches and the side branch. See Figure 2-1.

POINT SELECTION

From the **Selection** list, choose the points for the T-junction.

FRICTION SPECIFICATION

Enter a value or expression for the **Loss coefficient main branch** $K_{f,\text{main}}$ for the loss between main and junction pressure (the default is 0.1) and the **Loss coefficient side**

branch $K_{f,\,\text{side}}$ for the loss between the side pressure and junction pressure (the default is 1.2). Both are unitless numbers. See Figure 2-1.

The respective pressure drops are calculated as

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

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

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. If required, select the **Use weak constraints** check box.

The Non-Isothermal Pipe Flow Interface

The **Non-Isothermal Pipe Flow** interface (), found under the **Non-Isothermal Flow** branch () in the **Model Wizard**, has the equations, boundary conditions, and volume forces for modeling incompressible fluid flow and heat transfer in pipes.

When this interface is added, these default nodes are also added to the **Model Builder**—**Fluid**, **Pipe Properties**, **Pressure**, **Temperature**, and **Initial Values**. Right-click the **Non-Isothermal Pipe Flow** node to add other features that implement, for example, boundary conditions and volume forces.

Model	• Insulation of a Pipeline Section: Model Library path **Pipe_Flow_Module/Heat_Transfer/pipeline_insulation** • Geothermal Heating from a Pond Loop: Model Library path **Pipe_Flow_Module/Heat_Transfer/geothermal_heating** • Cooling of an Injection Mold: Model Library path **Pipe_Flow_Module/Heat_Transfer/mold_cooling**

2D *3D*	The interface is available in 3D on edges and 2D on boundaries.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `nipfl`.

EDGE OR BOUNDARY SELECTION

The default setting is to include **All edges** or **All boundaries** in the model. To choose specific edges or boundaries, select **Manual** from the **Selection** list.

FLUID MODEL

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

DEPENDENT VARIABLES

This 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)

For an explanation of the tangential velocity, see further in the section Theory for the Pipe Flow Interface.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. It controls the discretization (the element types used in the finite element formulation).

Select discretization options from the **Pressure** and **Tangential velocity** lists—**Quadratic** (the default), **Linear**, **Cubic**, or **Quartic**.

For each **Dependent variable** in the table under **Value types when using splitting of complex variables**, choose either a **Complex** or **Real Value type**. Click the cell to select from a drop-down list.

See Also

- Show More Physics Options
- Edge, Boundary, Point, and Pair Features for the Non-Isothermal Pipe Flow Interface
- Theory for the Pipe Flow Interface
- Theory for the Heat Transfer in Pipes Interface

Edge, Boundary, Point, and Pair Features for the Non-Isothermal Pipe Flow Interface

 2D  3D	The interface is available in 3D on edges and points, and in 2D on boundaries and points.

Because The Non-Isothermal Pipe Flow Interface is a multiphysics interface, it shares many features with other interfaces. These boundary, edge, point, and pair features are available as indicated (and listed in alphabetical order).

The Fluid node and Heat Transfer are described in this section.

These features are described for the **Reacting Pipe Flow** interface:

- Fluid Properties
- Pipe Properties

These features are described for the **Heat Transfer in Pipes** interface:

- External Film Resistance
- Internal Film Resistance
- Wall Heat Transfer
- Wall Layer

These features are described for the **Heat Transfer** interface in the *COMSOL Multiphysics User's Guide*:

- Heat Source
- Outflow
- Temperature

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

- Bend
- Contraction / Expansion
- Inlet

- No Flow
- Outlet
- Pressure
- Pump
- T-Junction
- Valve


See Also

In the *COMSOL Multiphysics User's Guide*:

- Volume Force is described for the **Laminar Flow** interface
- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs

Fluid

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

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the fluid.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) is **User defined** and is 1×10^3 kg/m^3. Enter a different value or expression as required.

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.

HEAT CONVECTION AND CONDUCTION

The default **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)), **Ratio of specific heats** γ (unitless), and **Thermal conductivity** k (SI unit: W/(m·K)) all use values **From material**. If **User defined** is selected, 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.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to apply the heat transfer properties.

MODEL INPUTS

This section contain fields and values that are inputs to expressions that define material properties. If such user-defined material models are added, the model inputs appear here. Initially, this section is empty.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) is **User defined** and is 1 x 10^3 kg/m^3. Enter a different value or expression as required.

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.

HEAT CONVECTION AND CONDUCTION

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

The Water Hammer Interface

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

When this interface is added, these default nodes are also added to the **Model Builder**—**Fluid Properties**, **Pipe Properties**, **Closed**, and **Initial Values**. Right-click the **Water Hammer** node to add other features.

Water Hammer: Model Library path
Pipe_Flow_Module/Verification_Models/water_hammer_verification

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `whtd`.

EDGE OR BOUNDARY SELECTION

The default setting is to include **All edges** or **All boundaries** in the model. To choose specific edges or boundaries, select **Manual** from the **Selection** list.

DEPENDENT VARIABLES

This 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 this section, click the **Show** button ( ) and select **Discretization**. It controls the discretization (the element types used in the finite element formulation).

Select discretization options from the **Pressure** and **Tangential velocity** lists—**Quadratic** (the default), **Linear**, **Cubic**, or **Quartic**.

For each **Dependent variable** in the table under **Value types when using splitting of complex variables**, choose either a **Complex** or **Real Value type**. Click the cell to select from a drop-down list.


See Also

- Show More Physics Options
- Edge, Boundary, Point, and Pair Features for the Water Hammer Interface
- Theory for the Pipe Flow Interface

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

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


2D


3D

The interface is available in 3D on edges and points, and in 2D on boundaries and points.

- Closed
- Fluid Properties
- Local Friction Loss
- Pipe Properties
- Pressure
- Velocity

The Initial Values node is described for the **Pipe Flow** interface.


See Also

In the *COMSOL Multiphysics User's Guide*:

- Volume Force is described for the **Laminar Flow** interface
- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs

Fluid Properties

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


Note

Volume Force is described for the **Laminar Flow** interface in the *COMSOL Multiphysics User's Guide*:

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to apply the fluid properties.

MODEL INPUTS

Edit input variables to the fluid-flow equations if required. For fluid flow, these are typically introduced when a material requiring inputs has been applied.

PHYSICAL PROPERTIES

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

The default **Density** ρ (SI unit: $\mathrm{kg/m^3}$) uses the value **From material**. Select **User defined** to 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**. Select **User defined** to enter a different value or expression.

Pipe Properties

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

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the pipe properties.

PIPE SHAPE


Note

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**.

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

FLOW RESISTANCE


Note

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** feature to add a lumped friction loss that depends on the velocity according to Equation 2-9.

POINT SELECTION

From the **Selection** list, choose the local friction loss points.

FRICTION SPECIFICATION

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

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Closed

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

POINT SELECTION

From the **Selection** list, choose the closed points.

Velocity

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

POINT SELECTION

From the **Selection** list, choose the velocity points.

VELOCITY

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

Pressure

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

POINT SELECTION

From the **Selection** list, choose the pressure 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**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Theory for the Pipe Flow Interface

The Pipe Flow Interface theory is described in this section:

- Flow Equations
- References for the Pipe Flow Interface

Flow Equations

The Pipe Flow interface calculates the pressure and velocity of an incompressible 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):

$$\rho\frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_D\frac{\rho}{2d_h}\mathbf{u}|\mathbf{u}| + \mathbf{F} \tag{2-3}$$

and

$$\frac{\partial A\rho}{\partial t} + \nabla\cdot(A\rho\mathbf{u}) = 0 \tag{2-4}$$

The second term on the right hand side in Equation 2-3 represents the pressure drop due to viscous shear. Here, $\mathbf{u}$ is the cross section averaged velocity (m/s), ρ the density (kg/m^3), p pressure (Pa), f_D (dimensionless) the Darcy friction factor (see Expressions for the Darcy Friction Factor) and $\mathbf{F}$ is a volume force term (N/m^3).

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

$$d_\mathrm{h} = \frac{4A}{Z} \tag{2-5}$$

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

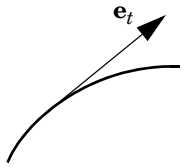


Figure 2-2: Unit tangent vector to the pipe axis.

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. Since all the velocity components normal to the pipe axis is assumed to be 0, the momentum balance in Equation 2-3 can be rewritten as

$$\mathbf{e}_t \cdot \left[\rho \frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_D \frac{\rho}{2 d_h} \mathbf{u}|\mathbf{u}| + \mathbf{F} \right] \tag{2-6}$$

and we can define the *tangential velocity* u as $\mathbf{u} = u\mathbf{e}_t$. The Physics *Pipe Flow* in COMSOL solves for the tangential velocity u. This is also the quantity the user specifies in for example inflow boundary conditions.

The following assumptions apply:

- The velocity profile is fully developed and does not change within a section.
- The cross section area is allowed to change between pipe segments.
- Empirical functions describe viscous losses
- Shocks are neglected
- The flow direction is in the direction of the pipe axis.

The Darcy friction factor in Equation 2-3 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).

$$f_D = f_D\left(\mathrm{Re}, \frac{e}{d_h}\right) \tag{2-7}$$

where

$$\mathrm{Re} = \frac{\rho u d_h}{\mu} \tag{2-8}$$

The physics interface automatically calculates f_D from one of the predefined expressions Equation 2-10 through Equation 2-16.

Additional Flow Resistances

Additional flow resistances can be added as point conditions between pipe segments—Bends, Valves, T-junctions, Contractions, and Expansions. These resistances give rise to abrupt (lumped) pressure drops in the points where they are added. The pressure drop is calculated as:

$$\Delta p = \frac{1}{2} K_f \rho u^2 \tag{2-9}$$

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 2-1: LOSS COEFFICIENTS

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

TABLE 2-1: LOSS COEFFICIENTS

FEATURE	DESCRIPTION	LOSS COEFFICIENT K
	Gradual, angle of convergence $\alpha < 22°$	$2.6\sin(\alpha)(1-\beta^2)^2$
	Gradual, angle of convergence $\alpha > 22°$	$(1-\beta^2)^2$

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).

Expressions for the Darcy Friction Factor

Newtonian 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 e/d:

$$f_D = 8\left[\left(\frac{8}{\mathrm{Re}}\right)^{12} + (A+B)^{-1.5}\right]^{1/12} \tag{2-10}$$

where

$$A = \left[-2.457\ln\left(\left(\frac{7}{\mathrm{Re}}\right)^{0.9} + 0.27(e/d)\right)\right]^{16} \tag{2-11}$$

$$B = \left(\frac{37530}{\mathrm{Re}}\right)^{16} \tag{2-12}$$

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

$$f_D = \frac{64}{\mathrm{Re}} \tag{2-13}$$

The Wood equation (Ref. 2) gives the friction factor for $4000 < \mathrm{Re} < 1\cdot10^7$ and $1\cdot10^{-5} < e/d < 0.04$, according to

$$f_D = 0.094(e/d)^{0.225} + 0.53(e/d) + 88(e/d)^{0.44}\cdot \mathrm{Re}^{a} \tag{2-14}$$

with

$$a = -1.62(e/d)^{0.134} \tag{2-15}$$

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

$$\sqrt{\frac{1}{f_D}} = -1.8\log_{10}\left(\left(\frac{e/d}{3.7}\right)^{1.11} + \left(\frac{6.9}{\mathrm{Re}}\right)\right) \tag{2-16}$$

It can be rewritten as

$$f_D = \frac{(1/1.8)^2}{\left(\log_{10}\left(\left(\frac{e/d}{3.7}\right)^{1.11} + \left(\frac{6.9}{\mathrm{Re}}\right)\right)\right)^2} \tag{2-17}$$

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

$$\sqrt{\frac{1}{f_D}} = -1.8\log_{10}\left(\frac{6.9}{\mathrm{Re}}\right) \tag{2-18}$$

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

$$\sqrt{\frac{1}{f_D}} = -1.8\log_{10}\left(\left(\frac{e/d}{3.7}\right)^{1.11}\right) = -2\log_{10}\left(\frac{e/d}{3.7}\right) \tag{2-19}$$

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

$$f_D = \frac{0.25}{\left(\log_{10}\left(\left(\frac{e/d}{3.7}\right) + \frac{5.74}{(\mathrm{Re})^{0.9}}\right)\right)^2} \tag{2-20}$$

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

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 Re<1000, COMSOL selects Stokes equation if it predicts a friction factor greater than equations for the turbulent regimes. This produces more accurate results that using the turbulent frictions 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 give pipe flow solution and change the friction model and recalculate, if necessary.

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

$$\mu = m\left(\frac{\partial\gamma}{\partial t}\right)^{n-1} \tag{2-21}$$

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):

$$f_D = \frac{64}{\mathrm{Re}_{\mathrm{MR}}} \tag{2-22}$$

$$\mathrm{Re}_{\mathrm{MR}} = \frac{\rho u^{(2-n)} d_h^n}{8^{(n-1)} m\left(\frac{3n+1}{4n}\right)^n} \tag{2-23}$$

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

$$f_D = 4\left(\frac{D(n)}{\mathrm{Re}_{\mathrm{MR}}}\right)^{\left(\frac{1}{3n+1}\right)} \tag{2-24}$$

where

$$D(n) = \frac{2^{(n+4)}}{7^{7n}}\left(\frac{4n}{3n+1}\right)^{3n^2} \tag{2-25}$$

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

$$\mathrm{Re}_{\mathrm{MR}} = \frac{6464n}{(3n+1)^2}(2+n)^{\left(\frac{2+n}{1+n}\right)} \tag{2-26}$$

predicts the critical Reynolds number.

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

$$\tau = \tau_B + \mu_B\left(\frac{\partial\gamma}{\partial t}\right) \quad \text{for } |\tau| > |\tau_B|$$
$$\left(\frac{\partial\gamma}{\partial t}\right) = 0 \qquad \text{for } |\tau| < |\tau_B| \tag{2-27}$$

The yield stress, τ_B (Pa), and the plastic viscosity, μ_B (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:

$$f_{D,\,\text{laminar}} = \frac{64}{\text{Re}_\text{B}} + \frac{10.67 + 0.1414\left(\frac{\text{He}}{\text{Re}_\text{B}}\right)^{1.143}}{\left(1 + 0.149\left(\frac{\text{He}}{\text{Re}_\text{B}}\right)^{1.16}\right)\text{Re}_\text{B}}\left(\frac{\text{He}}{\text{Re}_\text{B}}\right) \tag{2-28}$$

where

$$\text{Re}_\text{B} = \frac{\rho u d_h}{\mu_B} \tag{2-29}$$

and

$$\text{He} = \frac{\rho d_h \tau_B}{\mu_B^2} \tag{2-30}$$

is the Hedström number.

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

$$f_{D,\,\text{turbulent}} = 4 \cdot 10^{a_0}\text{Re}_\text{B}^{-0.193} \tag{2-31}$$

with

$$a_0 = -1.47(1 + \exp(-2.9 \cdot 10^{-5}\text{He})) \tag{2-32}$$

as well as an equation covering all flow regimes:

$$f_D = (f_{D,\,\text{laminar}}^{b} + f_{D,\,\text{turbulent}}^{b})^{\frac{1}{b}} \tag{2-33}$$

where

$$b = 1.7 + \frac{40000}{\text{Re}_B} \tag{2-34}$$


Note

The non-Newtonian friction models outlined above do not include any effects of wall roughness. As the laminar sub-layers tend to be thicker 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 2-2: SURFACE ROUGHNESS

TUBE MATERIAL	SURFACE ROUGHNESS (MM)
Drawn tubing	0.0015
Glass	0.0015
Thermoplastics	0.0015
Commercial steel and wrought iron	0.046
Steel, welded seamless	0.061
Asphalted cast iron	0.12
Galvanized iron	0.15
Cast-iron	0.26
Wood stave	0.18-0.9

TABLE 2-2: SURFACE ROUGHNESS

TUBE MATERIAL	SURFACE ROUGHNESS (MM)
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 equations 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," *Journal of 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," *Journal of the 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, "Transistion 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 Trhough 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.

Theory for the Water Hammer Interface

Flow Equations

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

THE CONTINUITY EQUATION

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

$$\frac{\partial A\rho}{\partial t} + \nabla \cdot (A\rho \mathbf{u}) = 0 \tag{2-35}$$

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

$$\frac{\partial A(p)\rho(p)}{\partial t} + \nabla \cdot (A(p)\rho(p)\mathbf{u}) = 0 \tag{2-36}$$

In a first order approximation, this equals

$$A_0\rho_0\left(\frac{1}{K_\rho} + \frac{1}{K_A}\right)\frac{\partial p}{\partial t} + \nabla \cdot (A_0\rho_0\mathbf{u}) = 0 \tag{2-37}$$

where K_ρ 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

$$\frac{1}{c^2} = \rho_0\left(\frac{1}{K_\rho} + \frac{1}{K_A}\right) \tag{2-38}$$

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

$$K_A = E\frac{w_{\text{th}}}{d_{\text{h}}}$$

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

$$K_A = \frac{E}{\zeta}\frac{w_{\text{th}}}{d_{\text{h}}} \tag{2-39}$$

where $\zeta = 1$ for pipe with zero axial stress (this is Korteweg's original formula for a pipe furbished 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:

$$\rho\frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_{\text{D}}\frac{\rho}{2d_{\text{h}}}\mathbf{u}|\mathbf{u}| + \mathbf{F} \tag{2-40}$$

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

$$A\frac{1}{c^2}\frac{\partial p}{\partial t} + \nabla \cdot (A\rho\mathbf{u}) = 0 \tag{2-41}$$

$$\rho\frac{\partial \mathbf{u}}{\partial t} = -\nabla p - f_{\text{D}}\frac{\rho}{2d_{\text{h}}}\mathbf{u}|\mathbf{u}| + \rho\mathbf{g} \tag{2-42}$$

BOUNDARY CONDITIONS

The available boundary conditions are Closed, Pressure, Velocity, and Additional Flow Resistances.

References for the Water Hammer interface

1. M. Ghidaoui, et al., *A Review of Water Hammer Theory and Practice*, Appl. Mech. Rev. 58, pp. 49–76, 2005

3

The Heat Transfer Branch

This physics, which is found under the **Heat Transfer** branch ( ) in the **Model Wizard**, 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** branch ( ) includes a number of subbranches to describe energy transport. One or more of them can be added from the Model Wizard.

Note

The Heat Transfer in Solids, Heat Transfer in Fluids (general convection and conduction, non-isothermal flow, and conjugate heat transfer), and Joule Heating interfaces all belong to the COMSOL Multiphysics base package. See The Heat Transfer Interface, The Joule Heating Interface and Theory for the Heat Transfer Interfaces in the *COMSOL Multiphysics User's Guide* for more information.

The **Heat Transfer in Pipes** interface ( ), found under the **Heat Transfer** branch ( ) in the **Model Wizard**, solves a temperature equation for a fluid transported in a pipe. The 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 Non-Isothermal 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.

Model

- Insulation of a Pipeline Section: Model Library path **Pipe_Flow_Module/Heat_Transfer/pipeline_insulation**
- Geothermal Heating from a Pond Loop: Model Library path **Pipe_Flow_Module/Heat_Transfer/geothermal_heating**
- Cooling of an Injection Mold: Model Library path **Pipe_Flow_Module/Heat_Transfer/mold_cooling**

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 (see mold_cooling above). The Heat Transfer in Pipes physics or the Non-Isothermal 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 goundflow models for porous media are better handled by the Subsurface Flow Module.

The Heat Transfer in Pipes Interface

The **Heat Transfer in Pipes** interface ( ), found under the **Heat Transfer** branch ( ) in the **Model Wizard**, has the equations and boundary conditions for modeling heat transfer for an incompressible fluid in one dimension.


2D


3D

The Heat Transfer in Pipes interface is available in 3D on edges and points, and in 2D on boundaries and points.

When this interface is added, these default nodes are also added to the **Model Builder**—**Heat Transfer**, **Pipe Properties**, **Temperature**, and **Initial Values**. Right-click the **Heat Transfer in Pipes** node to add other features.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `htp`.

EDGE OR BOUNDARY SELECTION

The default setting is to include **All edges** or **All boundaries** in the model. To choose specific edges or boundaries, select **Manual** from the **Selection** list.

DEPENDENT VARIABLES

This 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 this section, click the **Show** button ( ) and select **Discretization**. It controls the discretization (the element types used in the finite element formulation).

Select an option from the **Temperature** list—**Quadratic** (the default), **Linear**, **Cubic**, or **Quartic**. Specify the **Value type when using splitting of complex variables**—**Real** or **Complex** (the default).


See Also

- Show More Physics Options
- Edge, Boundary, Point, and Pair Features for the Heat Transfer in Pipes Interface
- Theory for the Heat Transfer in Pipes Interface

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


2D


3D

This interface is available in 3D on edges and points, and in 2D on boundaries and points.

The Heat Transfer in Pipes Interface has these boundary, edge, point, and pair features available and listed in alphabetical order:

- Heat Outflow
- Heat Source
- Heat Transfer
- Initial Values
- Pipe Properties (described for the **Pipe Flow** interface)
- Temperature
- Wall Heat Transfer

Heat Transfer

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

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to apply the heat transfer properties.

MODEL INPUTS

This section contain fields and values that are inputs to expressions that define material properties. If such user-defined material models are added, the model inputs appear here. Initially, this section is empty.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) uses the value **From material**. Select **User defined** to 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)**. If **User defined** is selected, enter a value for the **Tangential velocity** u (SI unit: m/s).

The default **Density** ρ (SI unit: kg/m^3), **Dynamic viscosity** μ (SI unit: Pa·s), **Heat capacity at constant pressure** C_p (SI unit: J/(kg·K)), **Ratio of specific heats** γ (unitless), and **Thermal conductivity** k (SI unit: W/(m·K)) all use values **From material**. If **User defined** is selected, 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 required—all heat sources within a domain contribute to the total heat source. Specify the heat source as W/m in the domain, as linear heat source.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to add the heat source.

HEAT SOURCE

Click the **General source** or **Linear source** button.

- If **General source** is selected, enter a value for the distributed heat source Q (SI unit: W/m^3).
- If **Linear source** ($Q = q_s \cdot T$) is selected, enter the **Linear heat source** q_s (SI unit: $W/(m^3 \cdot K)$).


Tip

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.


See Also

Stabilization Techniques in the *COMSOL Multiphysics Reference Guide*

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. Right-click to add Internal Film Resistance, Wall Layer, and External Film Resistance sub features. These compute heat transfer coefficients based on the section Theory for the Heat Transfer in Pipes Interface.


Note

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 surroundings fluid by forced or natural convection, add also External Film Resistance.

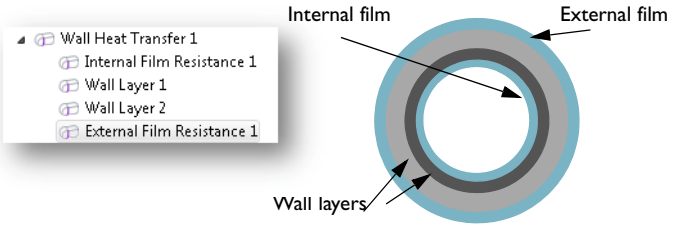


Figure 3-1: Internal and external films and the associated film coefficients (blue), and wall layers (dark and light grey).

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to apply the wall heat transfer properties.

HEAT TRANSFER MODEL

Select an option from the **External temperature** list—**User defined**, **Temperature (nipfl/nipfl)**, or **Temperature (pfl/pfl)**. If **User defined** is selected, enter a value for the **Temperature** T_{ext} (SI unit: K).

Enter a value or expression for the **Heat transfer coefficient** h (SI unit: W/(m^2·K). The default is 0.

Internal Film Resistance

Right-click the Wall Heat Transfer node to add the **Internal Film Resistance** node and define the film resistance.

Note	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).

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define the internal film resistance.

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. Select **User defined** to enter a user defined **Nusselt number** (unitless). The default is 3.66.

External Film Resistance

Right-click the Wall Heat Transfer node to add the **External Film Resistance** node and 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.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define the external film resistance.

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

If **External forced convection** or **External natural convection** is selected, the **External material** uses the **Domain material** by default.

Caution

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/m^3), and **Dynamic viscosity** μ (SI unit: Pa·s), all use values **From material**. If **User defined** is selected, 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.

Note	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 3-25), which may generate unexpected results or even unstable models. Also, make sure that the density correlation ρ(*T*) is smooth throughout the temperature interval used, or else β is discontinuous and may cause numerical instability.

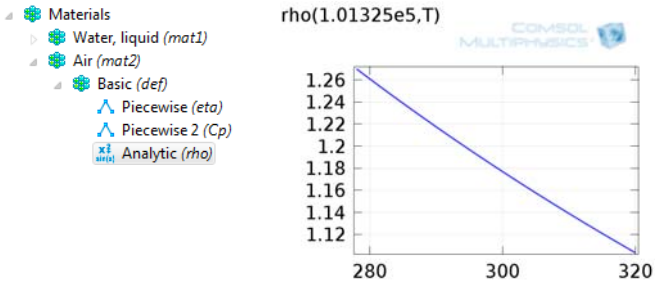


Figure 3-2: Before you use the Natural Convection option, it is good practice to select Density in the used material, and click Plot in the settings window. That way you can inspect the behavior and make sure it is smooth within the temperature interval you plan to use it.

User Defined

If **User defined** is selected:

- Enter the **Nusselt number** (unitless). The default is 3.66.
- The **External material** uses the **Domain material** by default
- The **Thermal conductivity** k (SI unit: W/(m·K)) takes its value **From material**. Select **User defined** to enter a different value or expression. The default is 0.
- Enter a value or expression for the **External pressure** p_{ext} (SI unit: Pa). The default is 1 atm).

Wall Layer

Right-click the Wall Heat Transfer node to add the **Wall Layer** node and define the thermal conductivity and wall thickness. You can add more layers.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define the wall layer.

SPECIFICATION

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

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

Initial Values

The **Initial Values** feature 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.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define initial values.

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.

POINT SELECTION

From the **Selection** list, choose the points to apply a temperature.

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**. If required, select the **Use weak constraints** check box.

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.

POINT SELECTION

From the **Selection** list, choose the points to apply the heat outflow.

Theory for the Heat Transfer in Pipes Interface

The Heat Transfer in Pipes Interface theory is described in this section:

- The Heat Transfer Equation
- Reference for the Heat Transfer in Pipes interface

The Heat Transfer Equation

This 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):

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

where ρ is the fluid density (kg/m^3), A is the pipe cross section area (m^2) available for flow, C_p (J/(kg·K)) is the heat capacity at constant pressure, T (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 (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 (W/m) represents a general heat source and Q_{wall} (W/m) represents external heat exchange through the pipe wall. Note that the Q_{wall} term is detailed below.

Wall Heat Transfer

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

$$Q_{\text{wall}} = (hZ)_{\text{eff}}(T_{\text{ext}} - T) \ \text{(W/m)} \qquad (3\text{-}2)$$

In Equation 3-2, $(hZ)_{\text{eff}}$ is an effective value of the heat transfer coefficient h (W/m^2·K) times the wall perimeter Z (m) of the pipe. T_{ext} (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 subfeatures to the Wall Heat Transfer feature. The subfeatures are:

- Internal Film Resistance
- Wall Layer
- External Film Resistance

T_{ext} in Equation 3-2 can be a constant, parameter, expression, or given by a temperature field computed by another physics interface, typically a 3D heat transfer physics interface. h is automatically calculated through film resistances and wall layers that are added as subnodes, see Equation 3-16 an 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 3-2) 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.

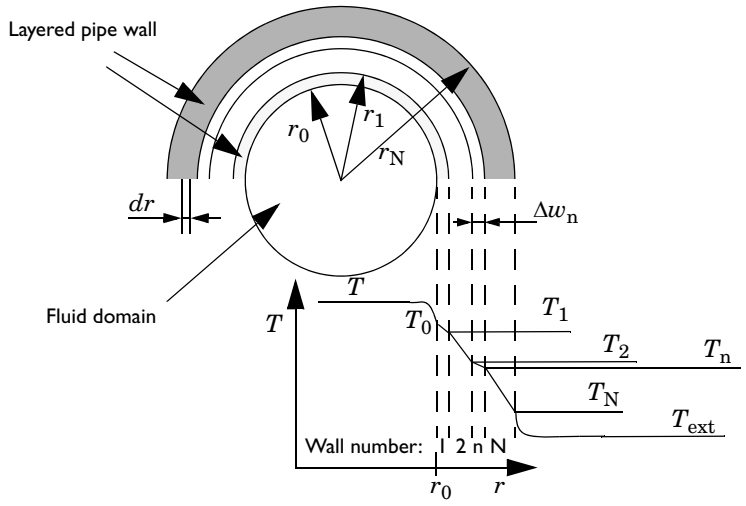


Figure 3-3: Temperature distribution across the pipe wall

- r_n (m) is the outer radius of wall n
- $w = r - r_0$ (m) a wall coordinate, starting at the inner radius r_0
- $\Delta w_n = r_n - r_{n-1}$ (m) the wall thickness of wall n
- Z_n (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 (W/(m^2, K)).
- k_n is the thermal conductivity (W/(m, K)) of wall n,

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

$$Q_0 = h_{int} A_Q \cdot (T - T_0)\,(\mathrm{W}) \tag{3-3}$$

Here, $A_Q = \Delta L 2\pi r_0$ (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).

$$Q_1 = \Delta L 2\pi r \cdot \left(-k_1 \frac{dT}{dr}\right) \tag{3-4}$$

Rearrange and integrate from r_0 to r_1.

$$\int_{T_0}^{T_1} dT = -\int_{r_0}^{r_0} \left(\frac{Q_1}{\Delta L 2\pi k_1} \cdot \frac{1}{r} \right) dr \tag{3-5}$$

Perform the integration

$$T_1 - T_0 = -\frac{Q_1}{\Delta L 2\pi k_1} \cdot \ln\left(\frac{r_1}{r_0}\right) \tag{3-6}$$

and rearrange

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

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 $Q_n = Q$.

$$\begin{aligned} Q &= h_{\text{int}} \Delta L 2\pi r_0 \cdot (T - T_0) \\ Q &= \frac{\Delta L 2\pi k_1}{\ln(r_1/r_0)} \cdot (T_0 - T_1) \\ Q &= \frac{\Delta L 2\pi k_2}{\ln(r_2/r_1)} \cdot (T_1 - T_2) \\ Q &= h_{\text{ext}} \Delta L 2\pi r_2 \cdot (T_2 - T_{\text{ext}}) \end{aligned} \tag{3-8}$$

Substituting

$$\Theta = \Delta L 2\pi\,, \tag{3-9}$$

and making a linear combination of the equations Equation 3-8 gives

$$Q = (hA_Q)_{\text{eff}} \cdot (T - T_{\text{ext}}) \tag{3-10}$$

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

$$(hA_Q)_{\text{eff}} = \frac{1}{\frac{1}{r_0 h_{\text{int}} \Theta} + \frac{\ln(r_1/r_0)}{k_1 \Theta} + \frac{\ln(r_2/r_1)}{k_2 \Theta} + \frac{1}{r_2 h_{\text{ext}} \Theta}} \tag{3-11}$$

For the general case with N wall layers this reads

$$(hA_Q)_{\text{eff}} = \frac{1}{\frac{1}{r_0 h_{\text{int}}\Theta} + \sum_{n=1}^{N}\left(\frac{\ln\left(\frac{r_n}{r_{n-1}}\right)}{k_n\Theta}\right) + \frac{1}{r_N h_{\text{ext}}\Theta}} \tag{3-12}$$

Now let

$$(hZ)_{\text{eff}} = \frac{(hA_Q)_{\text{eff}}}{\Delta L}, \tag{3-13}$$

where Z (m) is an average perimeter (circumference), of the pipe, taken over the thickness of the pipe walls. Combine Equation 3-9 and Equation 3-13 such a that

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

and insert in Equation 3-12:

$$(hZ)_{\text{eff}} = \frac{2\pi}{\frac{1}{r_0 h_{\text{int}}} + \frac{1}{r_N h_{\text{ext}}} + \sum_{n=1}^{N}\left(\frac{\ln\left(\frac{r_n}{r_{n-1}}\right)}{k_n}\right)} \tag{3-14}$$

For a circular pipe cross sections, this effective hZ in can now be used in Equation 3-2. 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 pipe
- the heat transfer through the wall is quasi-static. The latter means that the wall is assumed to immediately assume the equilibrium temperature distribution corresponding to T and T_{ext}. If this assumption would not be made, an auxiliary PDE across the wall coordinate would be required.

For square and rectangular pipe shapes the average conductance can be approximated by the simpler sum of resistances across a plane wall, which can be found in for example (Ref. 13):

$$(hZ)_{\text{eff}} = \frac{1}{\frac{1}{Z_0 h_{\text{int}}} + \frac{1}{Z_N h_{\text{ext}}} + \sum_{n=1}^{N}\frac{w_n}{k_n((Z_n + Z_{n-1})/2)}} \tag{3-15}$$

The film resistances can be calculated from

$$h = \mathrm{Nu}\frac{k}{d_\mathrm{h}} \tag{3-16}$$

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

$$d_\mathrm{h} = 4A/Z\,. \tag{3-17}$$

- The inner and outer film coefficients is evaluated at $(T+T_0)/2$ and $(T_\mathrm{N}+T_\mathrm{ext})/2$, respectively.
- The thermal conductivity k_n may be temperature dependent and is evaluated at $(T_\mathrm{n}+T_\mathrm{n\text{-}1})/2$
- To compute d_h in Equation 3-17 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 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 3-2 also can be applied for each individual wall layer:

$$Q_\mathrm{wall} = (hZ)_\mathrm{eff}(T_\mathrm{n+1} - T_\mathrm{n}) \tag{3-18}$$

Combining Equation 3-2, Equation 3-18 and Equation 3-14 or Equation 3-15 (depending on pipe shape) for each wall layer explicitly gives each T_n.

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

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

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

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 ($3000 < \mathrm{Re} < 6\cdot10^6$, $0.5 < \mathrm{Pr} < 2000$), the Gnielinski equation (Ref. 18) applies:

$$\mathrm{Nu}_{\mathrm{int}} = \frac{(f_D/8)(\mathrm{Re}-1000)\mathrm{Pr}}{1+12.7(f_D/8)^{1/2}(\mathrm{Pr}^{2/3}-1)} \tag{3-19}$$

Where Pr is the Prandtl number:

$$\mathrm{Pr} = \frac{C_p\mu}{k} \tag{3-20}$$

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 $\mathrm{Pr} > 0.2$, the Churchill and Bernstein (Ref. 19) correlation is used:

$$\mathrm{Nu}_{\mathrm{ext}} = 0.3 + \frac{0.62\mathrm{Re}^{1/2}\mathrm{Pr}^{1/3}}{[1+(0.4/\mathrm{Pr})^{2/3}]^{1/4}}[1+(\mathrm{Re}/282000)^{5/8}]^{4/5}. \tag{3-21}$$

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

$$\mathrm{Nu}_{\mathrm{ext}} = \left\{0.60 + \frac{0.387\mathrm{Ra}^{1/6}}{[1+(0.559/\mathrm{Pr})^{9/16}]^{8/27}}\right\}^2 \tag{3-22}$$

where the Rayleigh number is given by:

$$\mathrm{Ra} = \mathrm{PrGr} \tag{3-23}$$

and the Grashof number is:

$$\mathrm{Gr} = \frac{g\beta|T_{\mathrm{N}} - T_{\mathrm{ext}}|d^3}{\left(\frac{\mu}{\rho}\right)^2} \tag{3-24}$$

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

$$\beta = -\frac{1}{\rho}\left(\frac{\partial\rho}{\partial T}\right)\Big|_p \tag{3-25}$$

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

STABILIZATION OF THE HEAT TRANSFER EQUATION

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

See Also

Stabilization Techniques in the *COMSOL Multiphysics Reference Guide*

Reference for the Heat Transfer in Pipes interface

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

4

The Chemical Species Transport Branch

This chapter has information about the interfaces available under the **Chemical Species Transport** branch ( ) in the **Model Wizard**.

In this chapter:

The Transport of Diluted Species in Pipes Interface

The **Transport of Diluted Species in Pipes** interface ( ), found under the **Chemical Species Transport** branch ( ) in the **Model Wizard**, has the equations, boundary conditions, and reactions order to compute the concentration distribution of a solute in a dilute solution, considering diffusion, dispersion, convection, and chemical reactions. Note that the Reaction Pipe Flow physics uses this interface and couples it automatically to Pipe Flow and Heat Transfer in Pipes.

When this interface is added, these default nodes are also added to the **Model Builder**—**Convection and Diffusion**, **Concentration**, and **Initial Values**. Right-click the node to add other features that implement, for example, boundary conditions and volume forces.

 *2D*  *3D*	The interface is available in 3D on edges and points, and in 2D on boundaries and points.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `dsp`.

EDGE OR BOUNDARY SELECTION

The default setting is to include **All edges** or **All boundaries** in the model. To choose specific edges or boundaries, select **Manual** from the **Selection** list.

FLUID MODEL

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

DEPENDENT VARIABLES

This 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:

Note	To enable multiple species (more than one concentration), you need a license of either the Batteries and Fuel Cells Module, CFD Module, Chemical Reaction Engineering Module, Electrodeposition Module, Microfluidics Module, Subsurface flow Module, or the Corrosion Module.

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. It controls the discretization (the element types used in the finite element formulation).

Select discretization options from the **Concentration** list—**Quadratic** (the default), **Linear**, **Cubic**, or **Quartic**.

For each **Dependent variable** in the table under **Value types when using splitting of complex variables**, choose either a **Complex** or **Real Value type**. Click the cell to select from a drop-down list.

See Also	• Show More Physics Options • Edge, Boundary, Point, and Pair Features for the Transport of Diluted Species in Pipes Interface • Theory for the Transport of Diluted Species in Pipes Interface

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

The Transport of Diluted Species in Pipes Interface has these boundary, edge, point, and pair features are described in this section:

- Concentration
- Convection and Diffusion
- Initial Values
- Mass Outflow

- Pipe Properties (described for the **Pipe Flow** interface)
- Reactions
- Wall Mass Transfer


See Also

In the *COMSOL Multiphysics User's Guide*:

- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs

Convection and Diffusion

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

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

MODEL INPUTS

This section contain fields and values that are inputs to expressions that define material properties. If such user-defined material models are added, the model inputs appear here. Initially, this section is empty.

FLUID PROPERTIES

The default **Density** ρ (SI unit: $\mathrm{kg/m^3}$) uses the value **From material**. Select **User defined** to 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.

DIFFUSION COEFFICIENT

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

DISPERSION

Select a **Dispersion model**—**User defined** (the default), **Taylor (laminar, round pipes)**, or **Taylor (turbulent, round pipes)**. If **User defined** is selected, enter a value or expression for the **Dispersion coefficient** D_D (SI unit: m^2/s). The default is 0. Find more detail in the theory section Dispersion.

Initial Values

The **Initial Values** feature 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.

EDGE OR BOUNDARY SELECTION

Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the initial values.

INITIAL VALUES

Enter values or expressions for the initial value of the **Concentration** c (SI unit: mol/m^3). The default is 0.

Reactions

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

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define the reactions.

REACTIONS

Enter a **Reaction rate** R_c (SI unit: mol/(m^3·s)). The default is 0.

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 i specified to define the driving force.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define the wall mass transfer.

WALL MASS TRANSFER

For each species, enter a **Mass transfer coefficient** $k_{wall,c}$ (SI unit: m/s). The default is 0.

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.

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.

POINT SELECTION

From the **Selection** list, choose the points to define the concentration.

CONCENTRATION

Specify the **Concentration** c_{inc} (SI unit: mol/(m^3·s)) for each species. Enter a value or expression in the field for each species. The default is 0.

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. If required, select the **Use weak constraints** check box.

Mass Outflow

Use the **Mass Outflow** feature 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.

POINT SELECTION

From the **Selection** list, choose the points to define the mass outflow.

The Reacting Pipe Flow Interface

The **Reacting Pipe Flow** interface (), found under the **Chemical Species Transport** branch () in the **Model Wizard**, is a multiphysics interface that combines the three interfaces Pipe Flow, Heat Transfer in Pipes, and Transport of Diluted Species in Pipes. For the underlying theory, please refer to these respective sections in the documentation.

When this interface is added, these default nodes are also added to the **Model Builder**—**Fluid**, **Pipe Properties**, **Temperature**, **Pressure**, **Concentration**, and **Initial Values**. Right-click the node to add other features that implement, for example, boundary conditions and volume forces.

 *2D*  *3D*	The interface is available in 3D on edges and points, and in 2D on boundaries and points.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `rpfl`.

EDGE OR BOUNDARY SELECTION

The default setting is to include **All edges** or **All boundaries** in the model. To choose specific edges or boundaries, select **Manual** from the **Selection** list.

FLUID MODEL

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

DEPENDENT VARIABLES

This 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/m^3)

DISCRETIZATION

To display this section, click the **Show** button ( ) and select **Discretization**. It controls the discretization (the element types used in the finite element formulation).

Select discretization options from the **Pressure**, **Tangential velocity**, **Temperature**, and **Concentration** lists—**Quadratic** (the default), **Linear**, **Cubic**, or **Quartic**.

For each **Dependent variable** in the table under **Value types when using splitting of complex variables**, choose either a **Complex** or **Real Value type**. Click the cell to select from a drop-down list.

See Also

- Show More Physics Options
- Edge, Boundary, Point, and Pair Features for the Reacting Pipe Flow Interface

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

2D

3D

The interface is available in 3D on edges and points, and in 2D on boundaries and points.

Because the The Reacting Pipe Flow Interface is a multiphysics interface, many nodes are described for other interfaces. These boundary, edge, point, and pair features are described as indicated.

These features are described in this section:

- Initial Values
- Convection and Diffusion
- Fluid
- Fluid Properties
- Pipe Properties

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

- Bend
- Contraction / Expansion
- Inlet
- No Flow
- Outlet
- Pressure
- Pump
- T-Junction
- Valve


Note

Volume Force is described for the **Laminar Flow** interface in the *COMSOL Multiphysics User's Guide*:

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

- Concentration
- Mass Outflow
- Reactions
- Wall Mass Transfer

Pipe Properties

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

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the pipe properties.

PIPE SHAPE


Note

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 (unitless). The default is 0.

Initial Values

The **Initial Values** feature 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.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define initial values.

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/m^3). The default is 0 mol/m^3.

Fluid Properties

The **Fluid Properties** feature adds the momentum equations solved by the 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.


Note

Volume Force is described for the **Laminar Flow** interface in the *COMSOL Multiphysics User's Guide*:

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to apply the fluid properties.

MODEL INPUTS

Edit input variables to the fluid-flow equations if required. For fluid flow, these are typically introduced when a material requiring inputs has been applied.

FLUID PROPERTIES

The default **Density** ρ (SI unit: kg/m^3) is **User defined** and is 1 x 10^3 kg/m^3. Enter a different value or expression as required.

The 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.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define the convection and diffusion properties.

MODEL INPUTS

This section contain fields and values that are inputs to expressions that define material properties. If such user-defined material models are added, the model inputs appear here. Initially, this section is empty.

FLUID PROPERTIES

The default **Density** ρ (SI unit: $\mathrm{kg/m^3}$) is **User defined** and is 1×10^3 $\mathrm{kg/m^3}$. Enter a different value or expression as required.

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.

VELOCITY

Note

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: $\mathrm{m^2/s}$). The default is 1×10^{-9} $\mathrm{m^2/s}$.

DISPERSION

Select a **Dispersion model**—**User defined** (the default) or **Fan (laminar, round pipes)**. If **User defined** is selected, enter a value or expression for the **Dispersion coefficient** D_D (SI unit: $\mathrm{m^2/s}$). The default is 0. For detailed information about dispersion, see the theory section Dispersion.

Fluid

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

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the fluid.

FLUID PROPERTIES

The default **Density** ρ (SI unit: $\mathrm{kg/m^3}$) uses the value **From material**. Select **User defined** to 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

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

DISPERSION

Select a **Dispersion model**—**User defined** (the default), **Taylor (laminar, round pipes)**, or **Taylor (turbulent, round pipes)**. If **User defined** is selected, enter a value or expression for the **Dispersion coefficient** D_D (SI unit: $\mathrm{m^2/s}$). 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 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 an incompressible fluid flowing in a pipe is:

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

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

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 (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 uses the Taylor (Ref. 21) correlation for this second contribution, $D_{D,i}$ (m^2/s):

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

This expression is valid if:

$$\frac{L}{d} > 0.04\frac{ud}{D_i} \tag{4-3}$$

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

$$D_{D,\mathrm{i}} = 10.1\frac{d}{2}u\sqrt{\frac{f_D}{2}} \tag{4-4}$$

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

$$D_{D,\mathrm{i}} = k\frac{u^2d^2}{4D_i} \tag{4-5}$$

with k for Power-law fluids given by:

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

and k for Bingham plastic fluids:

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

The parameter ϕ_0 is

$$\phi_0 = \frac{r_0}{R} \tag{4-8}$$

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

$$r_0 = \frac{2L\tau_B}{\Delta p} \tag{4-9}$$

This can be rewritten as

$$\phi_0 = \frac{4\tau_B}{d\left|\nabla^T p\right|} \tag{4-10}$$

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 physics is numerically stabilized.

See Also

Stabilization Techniques in the *COMSOL Multiphysics Reference Guide*

5

The Acoustics Branch

This module has extra functionality for simulating acoustics in pipe networks, which is found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) in the **Model Wizard**.

In this chapter:

The Pipe Acoustics, Transient Interface

Note	This interface requires both the Pipe Flow Module and the Acoustics Module.

2D 3D	The interface is available in 3D on edges and points, and in 2D on boundaries and points.

The **Pipe Acoustics, Transient** interface ( ), found under the **Acoustics>Acoustic-Structure Interaction** branch ( ) in the **Model Wizard**, has the equations and boundary conditions for modeling the propagation of transient sound waves in flexible pipe systems. The equations are formulated in a general way to include the possibility of a stationary background flow.

When this interface is added, these default nodes are also added to the **Model Builder**—**Fluid Properties**, **Pipe Properties**, **Closed**, and **Initial Values**. Right-click the **Pipe Acoustics, Transient** node to add other features.

INTERFACE IDENTIFIER

The interface identifier is a text string that can be used to reference the respective physics interface if appropriate. Such situations could occur when coupling this interface to another physics interface, or when trying to identify and use variables defined by this physics interface, which is used to reach the fields and variables in expressions, for example. It can be changed to any unique string in the **Identifier** field.

The default identifier (for the first interface in the model) is `patd`.

EDGE OR BOUNDARY SELECTION

The default setting is to include **All edges** or **All boundaries** in the model. To choose specific edges or boundaries, select **Manual** from the **Selection** list.

DEPENDENT VARIABLES

This 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 this section, click the **Show** button ( ) and select **Discretization**. It controls the discretization (the element types used in the finite element formulation).

Select discretization options from the **Pressure** and **Tangential velocity** lists—**Linear**, **Quadratic**, **Cubic**, or **Quartic**. The defaults is quadratic for the pressure and linear for the tangential velocity.

For each **Dependent variable** in the table under **Value types when using splitting of complex variables**, choose either a **Complex** or **Real Value type**. Click the cell to select from a drop-down list.

See Also

- Show More Physics Options
- Edge, Boundary, Point, and Pair Features for the Pipe Acoustics, Transient Interface
- Theory for the Pipe Acoustics, Transient Interface

Edge, Boundary, Point, and Pair Features for the Pipe Acoustics, Transient Interface

The Pipe Acoustics, Transient Interface has these edge, boundary, point, and pair features available and listed in alphabetical order:

2D

3D

The interface is available in 3D on edges and points, and in 2D on boundaries and points.

- Closed

- Fluid Properties
- Initial Values
- Open
- Pipe Properties
- Pressure
- Velocity


See Also

In the *COMSOL Multiphysics User's Guide*:

- Volume Force is described for the **Laminar Flow** interface
- Continuity on Interior Boundaries
- Identity and Contact Pairs
- Specifying Boundary Conditions for Identity Pairs


Important

The links to the features described in the *COMSOL Multiphysics User's Guide* do not work in the PDF, only from the online help in COMSOL Multiphysics.


Tip

To locate and search all the documentation, in COMSOL, select **Help>Documentation** from the main menu and either enter a search term or look under a specific module in the documentation tree.

Initial Values

The **Initial Values** feature 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.

EDGE OR BOUNDARY SELECTION

From the **Selection** list, choose the edges or boundaries to define initial values.

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


Note

Volume Force is described for the **Laminar Flow** interface in the *COMSOL Multiphysics User's Guide*:

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the fluid properties.

MODEL INPUTS

Edit input variables to the fluid-flow equations if required. For fluid flow, these are typically introduced when a material requiring inputs has been applied.

BACKGROUND PROPERTIES

Enter a value or expression for the **Background velocity** u_0 (SI unit: m/s) and **Background pressure** p_0 (SI unit: Pa).

PHYSICAL PROPERTIES

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

The default **Density** ρ (SI unit: $\mathrm{kg/m^3}$) uses the value **From material**. Select **User defined** to enter a different value or expression.

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

Pipe Properties

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

EDGE OR BOUNDARY SELECTION


Note

For the default node no user selection is required. **All boundaries** or **All edges** is automatically selected.

When additional nodes are added, from the **Selection** list choose the boundaries or edges to define the pipe properties.

PIPE SHAPE

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

- If **Round** is selected, enter a value or expression for the **Inner diameter** d_i (SI unit: m). The default is 10 cm.
- If **Square** is selected, enter a value or expression for the **Inner width** w_i (SI unit: m). The default is 5 cm.
- If **Rectangular** is selected, 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 10cm).
- If **User defined** is selected, enter a value or expression for the **Cross sectional area** A_c (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 selected, also select a **Thickness** Δw—**Not set** (the default) or **User defined**. If **User defined** is selected, enter a value or expression.

- When **Anchored at one end** or **Anchored at both ends** is selected, also select a **Poisson's ratio** υ—**Not set** (the default) or **User defined**. If **User defined** is selected, enter a value or expression.

WALL DRAG FORCE

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

SWIRL CORRECTION

Enter a value or expression for β (unitless). The default is 1.

Closed

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

See Also

Theory for the Pipe Acoustics, Transient Interface Boundary Conditions

POINT SELECTION

From the **Selection** list, choose the closed points.

Open

Use the **Open** feature to model an open pipe end where waves propagate out of the system without any reflections. The boundary condition prescribes the velocity given for plane waves moving in an infinitely long pipe of constant cross section area A. The relation between the pressure p and tangential velocity u is given by

$$\frac{u}{p} = \frac{1}{c^2}\frac{1}{\rho}\left((\beta-1)u_0 + c\sqrt{\beta(\beta-1)\left(\frac{u_0}{c}\right)^2 + (1+\beta_A p_0)}\right)$$

where u_0 is the background tangential flow, c is the wave speed, ρ is the fluid density, β_A is the cross sectional area compressibility, and β is the swirl correction factor.

See Also

Theory for the Pipe Acoustics, Transient Interface Boundary Conditions

POINT SELECTION

From the **Selection** list, choose the open points.


Note

The wave speed c may be different from the speed of sound c_s and depends on the elastic properties of the pipe structure. It is defined in Equation 5-3 in the Governing Equations section.

The wave speed may be evaluated as `sqrt(1/patd.invc2)` during the analysis and results stage.

Pressure

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


See Also

Theory for the Pipe Acoustics, Transient Interface Boundary Conditions

POINT SELECTION

From the **Selection** list choose the points that represent the pressure.

PRESSURE

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

CONSTRAINT SETTINGS

To display this section, click the **Show** button ( ) and select **Advanced Physics Options**. Select a **Constraint type**—**Bidirectional, symmetric** or **Unidirectional**. If required, select the **Use weak constraints** check box.

Velocity

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


See Also

Theory for the Pipe Acoustics, Transient Interface Boundary Conditions

POINT SELECTION

From the **Selection** list, choose the velocity points.

VELOCITY

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

Theory for the Pipe Acoustics, Transient Interface

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 model 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 Interface theory is discussed in this section:

- Governing Equations
- Theory for the Pipe Acoustics, Transient Interface Boundary Conditions
- Solving Transient Problems
- References for the Pipe Acoustics, Transient Interface


Note

The **Pipe Acoustics, Transient** interface requires both the Pipe Flow Module and the Acoustics Module.

Governing Equations

The continuity equation derived for a control volume is given by

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

and the corresponding momentum balance equation is

$$\frac{\partial(\rho A\mathbf{u})}{\partial t} + \nabla(\rho A\beta\mathbf{u}^2) = -\nabla(Ap) + \mathbf{f}_{\text{drag}}Z + A\mathbf{F} \tag{5-2}$$

where Z is the inner circumference of the pipe and $A = A(x,p,...)$ is the inner wetted cross sectional area. $\mathbf{u}$ is the area-averaged mean velocity, and it is also defined in the tangential direction $\mathbf{u} = u\mathbf{e}_t$, p is the mean pressure along the pipe, $\mathbf{f}_{\text{drag}}$ is the 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 swirl-correction factor relating the mean of the squared total velocity to the square of the mean velocity. Such that

$$u = \frac{1}{A}\int \tilde{\mathbf{u}} \cdot \mathbf{dA} \qquad p = \frac{1}{A}\int \tilde{p} dA \qquad \beta = \left(\frac{1}{A}\left(\int \tilde{u}^2 dA\right)\right) / u^2$$

where

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

are here the local parameters. Again p and u are the 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 (0th) 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 may be changed by external factors such as heating or structural deformation, thus the time dependency. The 1st order terms represent small perturbations on top of the background values (0th order). They are valid for

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

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

$$\rho_1 = \rho - \rho_0 = (p - p_0)\left[\frac{\partial \rho}{\partial p}\right]_s\Bigg|_0$$

$$A_1 = A - A_0 = (p - p_0)\left[\frac{\partial A}{\partial p}\right]_s\Bigg|_0$$

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

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

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

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 section A_0 due to changes in the inner fluid pressure.

Inserting the above expansions into the governing equations (Equation 5-1 and Equation 5-2) and retaining only 1st order terms yield the pipe acoustics equations including background flow. These are (after some manipulation and dropping the suffix 1 on the acoustic variations):

$$\begin{aligned} A_0 \frac{1}{c^2}\frac{\partial p}{\partial t} + \nabla \cdot \left(A_0 \rho_0 \left(\mathbf{u}_1 + \frac{\mathbf{u}_0}{\rho_0 c^2} p \right) \right) &= 0 \\ \rho_0 A_0 \left(\frac{\partial \mathbf{u}}{\partial t} + \frac{\mathbf{u}_0}{\rho_0 c^2}\frac{\partial p}{\partial t} \right) + \nabla \left(A_0 \beta \frac{\mathbf{u}_0^2}{c^2} p + 2\rho_0 A_0 \beta \mathbf{u}_0 \mathbf{u} \right) & \\ + \nabla((A_0 p)(1 + p_0 \beta_A)) + \mathbf{f}_{\text{drag}} Z &= 0 \\ \frac{1}{c^2} = \rho_0(\beta_0 + \beta_A) = \rho_0\left(\frac{1}{K_0} + \frac{1}{K_A}\right) &= \frac{1}{c_s^2} + \frac{\rho_0}{K_A} \end{aligned} \tag{5-3}$$

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).

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

$$A\frac{1}{c^2}\frac{\partial p}{\partial t} + \nabla\left(A\rho\left(u + \frac{u_0}{\rho c^2}p\right)\right)\cdot \mathbf{e}_t = 0$$

$$\rho A\left(\frac{\partial u}{\partial t} + \frac{u_0}{\rho c^2}\frac{\partial p}{\partial t}\right) + \nabla\left(A\beta\frac{u_0^2}{c^2}p + 2\rho A\beta u_0 u\right)\cdot \mathbf{e}_t \tag{5-4}$$
$$+ \nabla(Ap(1+p_0\beta_A))\cdot \mathbf{e}_t + \tau_w Z + A(\mathbf{F}\cdot \mathbf{e}_t) = 0$$

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

where τ_w is the tangential wall drag force (SI unit: N/m^2) and $\mathbf{F}$ is a volume force (SI unit: N/m^3).

Theory for the Pipe Acoustics, Transient Interface Boundary 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

$$p = p_{in}$$

and the Velocity condition

$$u = u_{in}$$

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

$$u = 0$$

this corresponds to the sound-hard wall condition in pressure acoustics.

The final condition to specify is the Open condition. In this case it is assumed that the pipe continues with constant cross section A. This requires specifying as the acoustic waves are, by design, always normal to the pipe ends, for example, the velocity as a function of the pressure. In order to define this, the dispersion relation and the relation between p and u for a plane wave needs to be determined.

To determine these relations insert the assumed plane wave form

$$p = \mathrm{Re}(\tilde{p}e^{i(\omega t - kx)})$$
$$u = \mathrm{Re}(\tilde{u}e^{i(\omega t - kx)})$$

into the governing Equation 5-4 and solve for the desired relations. After some manipulation this results in

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

with the dispersion relation

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

such that the relation

$$\frac{u}{p} = \frac{1}{c^2}\frac{1}{\rho}\left((\beta-1)u_0 + c\sqrt{\beta(\beta-1)\left(\frac{u_0}{c}\right)^2 + (1 + \beta_A p_0)}\right)$$

must hold at an open boundary.

Solving Transient Problems

When solving transient acoustic problems where the wave shape is not necessarily harmonic it may 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. For transient acoustic problems it is defined as

$$\mathrm{CFL} = \frac{dx}{c \cdot dt} = 0{,}2$$

This conditions restricts any acoustic disturbances to propagate more than 20 % of the mesh size dx during one time step dt.


Model

For an example of a model where the CFL condition is used see Water Hammer: Model Library path **Pipe_Flow_Module/Verification_Models/water_hammer_verification**.

References for the Pipe Acoustics, Transient Interface

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.

6

Materials

This chapter describes how to use the material databases.

In this chapter:

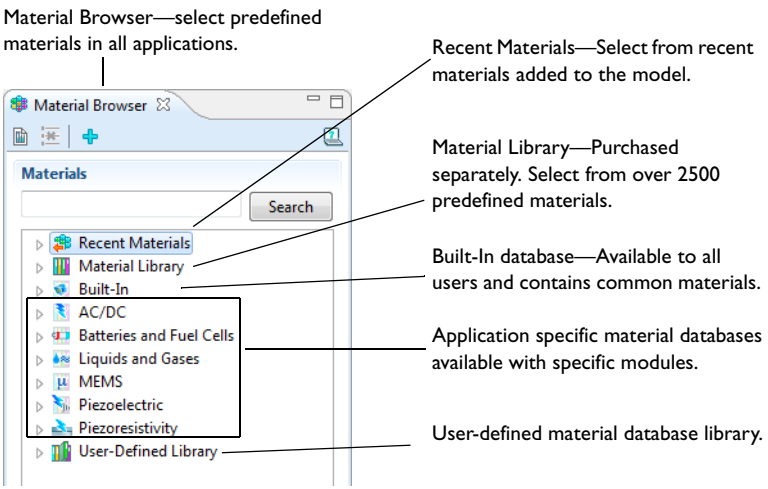
Material Library and Databases

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

Note	For detailed information about all the other material databases and the separately purchased Material Library, see Materials in the *COMSOL Multiphysics User's Guide*.

In this section:

About the Material Databases



All COMSOL modules have predefined material data available to build models. The most extensive material data is contained in the separately purchased Material Library,

but all modules contain commonly used or module-specific materials. For example, the *Built-In* database is available to all users but the *MEMS* database is included with the MEMS Module and Structural Mechanics Module. Also create custom materials and material libraries by researching and entering material properties.

All the material databases (including the Material Library) are accessed from the **Material Browser**. These databases are briefly described below.

RECENT MATERIALS

From the **Recent Materials** folder (), select from a list of recently used materials, with the most recent at the top. This folder is available after the first time a material is added to a model.

MATERIAL LIBRARY

An optional add-on database, the **Material Library** (), contains data for over 2500 materials and 20,000 property functions.

BUILT-IN

Included with COMSOL Multiphysics, the **Built-In** database () contains common solid materials with electrical, structural, and thermal properties.


See Also

Predefined Built-In Materials for all COMSOL Modules in the *COMSOL Multiphysics User's Guide*

AC/DC

Included in the AC/DC Module, the **AC/DC** database () has electric properties for some magnetic and conductive materials.

BATTERIES AND FUEL CELLS

Included in the Batteries & Fuel Cells Module, the **Batteries and Fuel Cells** database () includes properties for electrolytes and electrode reactions for certain battery chemistries.

LIQUIDS AND GASES

Included in the Acoustics Module, CFD Module, Chemical Reaction Engineering Module, Heat Transfer Module, MEMS Module, Pipe Flow Module, and Subsurface

Flow Module, the **Liquids and Gases** database ( ) includes transport properties and surface tension data for liquid/gas and liquid/liquid interfaces.

MEMS

Included in the MEMS Module and Structural Mechanics Module, the **MEMS** database ( ) has properties for MEMS materials—metals, semiconductors, insulators, and polymers.

PIEZOELECTRIC

Included in the Acoustics Module, MEMS Module, and Structural Mechanics Module, the **Piezoelectric** database ( ) has properties for piezoelectric materials.

PIEZORESISTIVITY

Included in the MEMS Module, the **Piezoresistivity** database ( ) has properties for piezoresistive materials, including p-Silicon and n-Silicon materials.

USER-DEFINED LIBRARY

The **User-Defined Library** folder ( ) is where user-defined materials databases (libraries) are created. When any new database is created, this also displays in the **Material Browser**.

Important	The materials databases shipped with COMSOL Multiphysics are read-only. This includes the Material Library and any materials shipped with the optional modules.

See Also	Creating Your Own User-Defined Libraries in the *COMSOL Multiphysics User's Guide*

About Using Materials in COMSOL

USING THE MATERIALS IN THE PHYSICS SETTINGS

The physics set-up in a model is determined by a combination of settings in the **Materials** and physics interface nodes. When the first material is added to a model, COMSOL automatically assigns that material to the entire geometry. Different geometric entities can have different materials. The following example uses the

heat_sink.mph model file contained in the Heat Transfer Module and CFD Module Model Libraries.

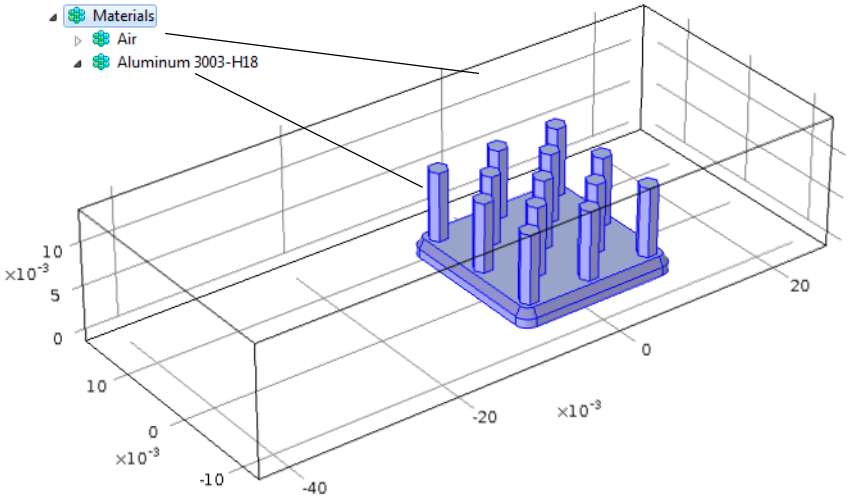


Figure 6-1: Assigning materials to a heat sink model. Air is assigned as the material to the box surrounding the heat sink, and aluminum to the heat sink itself.

If a geometry consists of a heat sink in a container, **Air** can be assigned as the material in the container surrounding the heat sink and **Aluminum** as the heat sink material itself (see Figure 6-1). The **Conjugate Heat Transfer** interface, selected during model set-up, has a **Fluid** flow model, defined in the box surrounding the heat sink, and a **Heat Transfer** model, defined in both the aluminum heat sink and in the air box. The **Heat Transfer in Solids 1** settings use the material properties associated to the **Aluminum 3003-H18** materials node, and the **Fluid 1** settings define the flow using the **Air** material properties. The other nodes under **Conjugate Heat Transfer** define the initial and boundary conditions.

All physics interface properties automatically use the correct **Materials'** properties when the default **From material** setting is used. This means that one node can be used to define the physics across several domains with different materials; COMSOL then uses the material properties from the different materials to define the physics in each domain. If material properties are missing, the **Material Contents** section on the

Materials page displays a stop icon ( ) to warn about the missing properties and a warning icon ( ) if the property exists but its value is undefined.


See Also

The Material Page in the *COMSOL Multiphysics User's Guide*

There are also some physics interface features where you can explicitly select a material from which material properties are retrieved. The default setting is then always to use the **Domain material** on each domain (that is, the materials defined on the same domains as the physics interface). In addition to the **Domain Material**, you can select any other material that is present in the model, regardless of its selection. The selected material's properties are then applied to all domains in the feature's selection.

EVALUATING AND PLOTTING MATERIAL PROPERTIES

You can access the material properties for evaluation and plotting like other variables in a model using the variable naming conventions and scoping mechanisms:

- To access a material property throughout the model (across several materials) and not just in a specific material, use the special material container `root.material`. For example, `root.material.rho` is the density ρ as defined by the materials in each domain in the geometry. For plotting, you can type the expression `material.rho` to create a plot that shows the density of all materials.


Note

If you use a temperature-dependent material, each material contribution asks for a special model input. For example, `rho(T)` in a material `mat1` asks for `root.mat1.def.T`, and you need to define this variable (`T`) manually—if the temperature is not available as a dependent variable—to make the density variable work.

- To access a material property from a specific material, you need to know the tags for the material and the property group. Typically, for the first material (Material 1) the tag is `mat1` and most properties reside in the default **Basic** property group with the tag `def`. The variable names appear in the **Variable** column in the table under **Output properties** in the settings window for the property group; for example, `Cp` for the heat capacity at constant pressure. The syntax for referencing the heat capacity at constant pressure in Material 1 is then `mat1.def.Cp`. Some properties are anisotropic tensors, and each of the components can be accessed, such as

`mat1.def.k11`, `mat1.def.k12`, and so on, for the thermal conductivity. For material properties that are functions, call these with input arguments such as `mat1.def.rho(pA,T)` where `pA` and `T` are numerical values or variables representing the absolute pressure and the temperature, respectively. The functions can be plotted directly from the function nodes' settings window by first specifying suitable ranges for the input arguments.

- Many physics interfaces also define variables for the material properties that they use. For example, `solid.rho` is the density in the Solid Mechanics interface and is equal to the density in a material when it is used in the domains where the Solid Mechanics interface is active. If you define the density in the Solid Mechanics interface using another value, `solid.rho` represents that value and not the density of the material. If you use the density from the material everywhere in the model, `solid.rho` and `material.rho` are identical.

Opening the Material Browser


Note

When using the **Material Browser**, the words *window* and *page* are interchangeable. For simplicity, the instructions refer only to the **Material Browser**.

1 Open or create a model file.

2 From the **View** menu choose **Material Browser** or right-click the **Materials** node and choose **Open Material Browser**.

The **Material Browser** opens by default in the same position as the settings window.

3 Under **Material Selection**, search or browse for materials.

- Enter a **Search** term to find a specific material by name, UNS number (Material Library materials only), or DIN number (Material Library materials only). If the search is successful, a list of filtered databases containing that material displays under **Material Selection**.


Tip

To clear the search field and browse, delete the search term and click **Search** to reload all the databases.

- Click to open each database and browse for a specific material by class (for example, in the Material Library) or physics module (for example, MEMS Materials).

Always review the material properties to confirm they are applicable for the model. For example, **Air** provides temperature-dependent properties that are valid at pressures around 1 atm.

4 When the material is located, right-click to **Add Material to Model**.

A node with the material name is added to the **Model Builder** and the **Material** page opens.

Using Material Properties

For detailed instructions, see Adding Predefined Materials and Material Properties Reference in the *COMSOL Multiphysics User's Guide*.

Liquids and Gases Materials Database


Model

For an example of the Liquids and Gases materials database, see Discharging Tank: Model Library path **Pipe_Flow_Module/Tutorial_Models/discharging_tank**.

In this section:

Liquids and Gases Materials

The Liquids and Gases materials database contains thermal and fluid dynamic properties for a set of common liquids. All properties are given as functions of temperature and at atmospheric pressure, except the density, which for gases is also a function of the local pressure. The database also contains surface and interface tensions

for a selected set of liquid/gas and liquid/liquid systems. All functions are based on data collected from scientific publications.

TABLE 6-1: LIQUIDS AND GASES MATERIALS

GROUP	MATERIAL
Gases References 1, 2, 7, 8, 11, and 12	Air
	Nitrogen
	Oxygen
	Carbon dioxide
	Humid air
	Hydrogen
	Helium
	Steam
	Propane
	Ethanol vapor
	Diethyl ether vapor
	Freon12 vapor
	SiF4
Liquids References 2, 3, 4, 5, 6, 7, 9, and 10	Engine oil
	Ethanol
	Diethyl ether
	Ethylene glycol
	Gasoline
	Glycerol
	Heptane
	Mercury
	Toluene
	Transformer oil
	Water

DEFAULT GRAPHICS WINDOW APPEARANCE SETTINGS

3D

The **MEMS Materials** database has data for these materials and a default appearance for 3D models is applied to each material as indicated.

See Also

See Working on the Material Page in the *COMSOL Multiphysics User's Guide* for more information about customizing the material's appearance in the Graphics window.

TABLE 6-2: MATERIALS 3D MODEL DEFAULT APPEARANCE SETTINGS

MATERIAL	DEFAULT FAMILY	DEFAULT LIGHTING MODEL	CUSTOM DEFAULT SETTINGS					
			NORMAL VECTOR NOISE SCALE	NORMAL VECTOR NOISE FREQUENCY	DIFFUSE AND AMBIENT COLOR OPACITY	SPECULAR EXPONENT	REFLECTANCE AT NORMAL INCIDENCE	SURFACE ROUGHNESS
All gases	Air	Simple	0.08	3	0.1	-	-	-
All liquids (except Engine oil, Mercury, and Transformer oil)	Water	Cook-Torrance	0.2	0.2	0.2	-	0.7	0.05
Engine oil	Custom	Cook-Torrance	0.2	0.2	0.2	-	0.7	0.05
Mercury	Custom	Cook-Torrance	0	1	1	-	0.9	0.1
Transformer oil	Plastic	Blinn-Phong	0	1	1	64	-	-

References for the Liquids and Gases Material Database

1. *ASHRAE Handbook of Fundamentals*, American Society of Heating, Refrigerating and Air Conditioning Engineers, 1993.

2. E. R. G. Eckert and M. Drake, Jr., *Analysis of Heat and Mass Transfer*, Hemisphere Publishing, 1987.

3. H. Kashiwagi, T. Hashimoto, Y. Tanaka, H. Kubota, and T. Makita, "Thermal Conductivity and Density of Toluene in the Temperature Range 273-373K at Pressures up to 250 MPa," *Int. J. Thermophys.*, vol. 3, no. 3, pp. 201–215, 1982.

4. C. A. Nieto de Castro, S.F.Y. Li, A. Nagashima, R.D. Trengove, and W.A. Wakeham, "Standard Reference Data for the Thermal Conductivity of Liquids," *J. Phys. Chem. Ref. Data*, vol. 15, no. 3, pp. 1073–1086, 1986.

5. B.E. Poling, J.M. Prausnitz, and J.P. O'Connell, *The Properties of Gases and Liquids*, 5th ed., McGraw-Hill, 2001.

6. C.F. Spencer and B.A. Adler, "A Critical Review of Equations for Predicting Saturated Liquid Density," *J. Chem. Eng. Data*, vol. 23, no. 1, pp. 82–88, 1978.

7. N.B.Vargnaftik, *Tables of Thermophysical Properties of Liquids and Gases*, 2nd ed., Hemisphere Publishing, 1975.

8. R.C.Weast (editor), *CRC Handbook of Chemistry and Physics*, 69th ed., CRC Press, 1988.

9. M. Zabransky and V. Ruzicka, Jr., "Heat Capacity of Liquid n-Heptane Converted to the International Temperature Scale of 1990," *Phys. Chem. Ref. Data*, vol. 23, no. 1, pp. 55–61, 1994.

10. M. Zabransky, V. Ruzicka, Jr., and E.S. Domalski, "Heat Capacity of Liquids: Critical Review and Recommended Values. Supplement I," *J. Phys. Chem. Ref. Data*, vol. 30, no. 5, pp. 1199–1397, 2002.

11. W. Wagner, and H-J Kretzschmar, *International Steam Tables*, 2nd ed., Springer, 2008.

12. F.P. Incropera and D.P. DeWitt, *Fundamentals of Heat and Mass Transfer*, Fifth ed. John Wiley & Sons, 2002.

Index