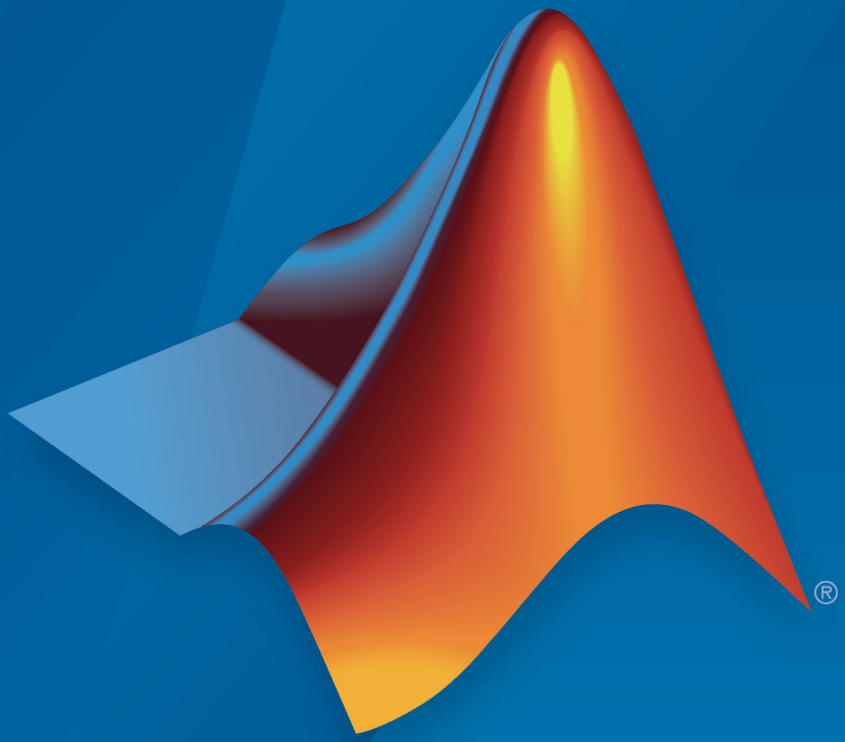
Simscape™ Fluids™

Reference



MATLAB® & SIMULINK®

R2018b

MathWorks®

How to Contact MathWorks

Latest news: `www.mathworks.com`

Sales and services: `www.mathworks.com/sales_and_services`

User community: `www.mathworks.com/matlabcentral`

Technical support: `www.mathworks.com/support/contact_us`

Phone: 508-647-7000

The MathWorks, Inc.
3 Apple Hill Drive
Natick, MA 01760-2098

Simscape™ Fluids™ Reference

© COPYRIGHT 2006–2018 by The MathWorks, Inc.

The software described in this document is furnished under a license agreement. The software may be used or copied only under the terms of the license agreement. No part of this manual may be photocopied or reproduced in any form without prior written consent from The MathWorks, Inc.

FEDERAL ACQUISITION: This provision applies to all acquisitions of the Program and Documentation by, for, or through the federal government of the United States. By accepting delivery of the Program or Documentation, the government hereby agrees that this software or documentation qualifies as commercial computer software or commercial computer software documentation as such terms are used or defined in FAR 12.212, DFARS Part 227.72, and DFARS 252.227-7014. Accordingly, the terms and conditions of this Agreement and only those rights specified in this Agreement, shall pertain to and govern the use, modification, reproduction, release, performance, display, and disclosure of the Program and Documentation by the federal government (or other entity acquiring for or through the federal government) and shall supersede any conflicting contractual terms or conditions. If this License fails to meet the government's needs or is inconsistent in any respect with federal procurement law, the government agrees to return the Program and Documentation, unused, to The MathWorks, Inc.

Trademarks

MATLAB and Simulink are registered trademarks of The MathWorks, Inc. See `www.mathworks.com/trademarks` for a list of additional trademarks. Other product or brand names may be trademarks or registered trademarks of their respective holders.

Patents

MathWorks products are protected by one or more U.S. patents. Please see `www.mathworks.com/patents` for more information.

Revision History

March 2006	Online only	New for Version 1.0 (Release 2006a+)
September 2006	Online only	Revised for Version 1.1 (Release 2006b)
March 2007	Online only	Revised for Version 1.2 (Release 2007a)
September 2007	Online only	Revised for Version 1.2.1 (Release 2007b)
March 2008	Online only	Revised for Version 1.3 (Release 2008a)
October 2008	Online only	Revised for Version 1.4 (Release 2008b)
March 2009	Online only	Revised for Version 1.5 (Release 2009a)
September 2009	Online only	Revised for Version 1.6 (Release 2009b)
March 2010	Online only	Revised for Version 1.7 (Release 2010a)
September 2010	Online only	Revised for Version 1.8 (Release 2010b)
April 2011	Online only	Revised for Version 1.9 (Release 2011a)
September 2011	Online only	Revised for Version 1.10 (Release 2011b)
March 2012	Online only	Revised for Version 1.10.1 (Release 2012a)
September 2012	Online only	Revised for Version 1.11 (Release 2012b)
March 2013	Online only	Revised for Version 1.12 (Release 2013a)
September 2013	Online only	Revised for Version 1.13 (Release 2013b)
March 2014	Online only	Revised for Version 1.14 (Release 2014a)
October 2014	Online only	Revised for Version 1.15 (Release 2014b)
March 2015	Online only	Revised for Version 1.16 (Release 2015a)
September 2015	Online only	Revised for Version 1.17 (Release 2015b)
March 2016	Online only	Revised for Version 2.0 (Release 2016a) (Renamed from *SimHydraulics® Reference*)
September 2016	Online only	Revised for Version 2.1 (Release R2016b)
March 2017	Online only	Revised for Version 2.2 (Release R2017a)
September 2017	Online only	Revised for Version 2.3 (Release R2017b)
March 2018	Online only	Revised for Version 2.4 (Release R2018a)
September 2018	Online only	Revised for Version 2.5 (Release R2018b)

Contents

1 Blocks — Alphabetical List

A Bibliography

Glossary

Blocks — Alphabetical List

2-Position Valve Actuator

Actuator for two-position valves

Library

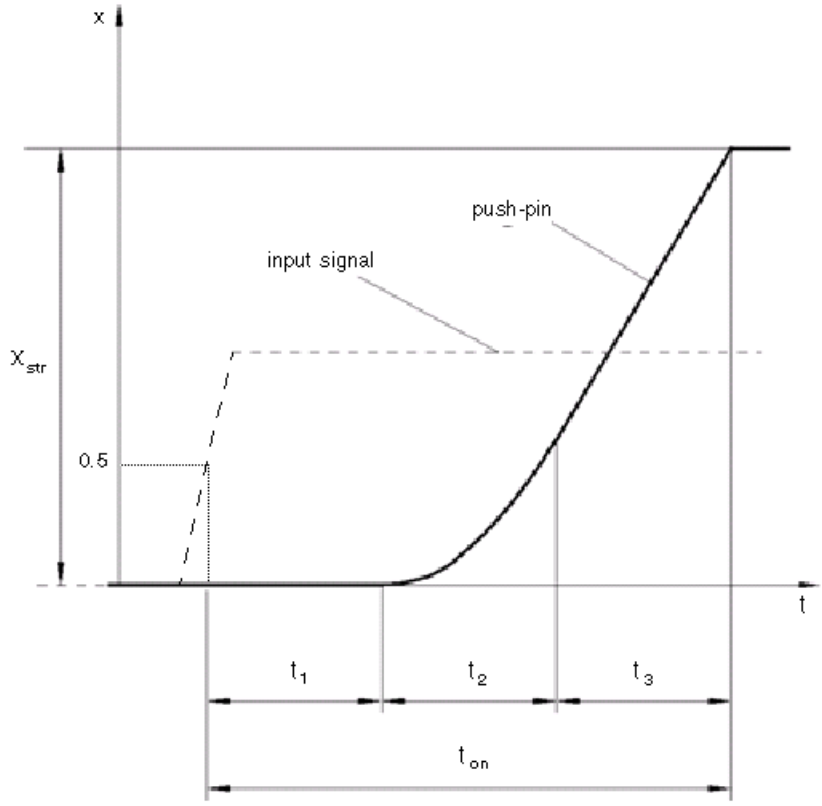
Valve Actuators

Description

The 2-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a two-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The key parameters are the stroke, switch-on, and switch-off times.

The block accepts a physical input signal and produces a physical output signal that can be associated with a mechanical translational or rotational push-pin motion. Connect the block output to the directional valve control port.

The actuator is represented as an ideal transducer, where output does not depend on the load exerted on the push-pin and the push-pin motion profile remains the same under any loading conditions. The motion profile represents a typical transition curve for electromagnetic actuators and is shown in the following figure:



The push-pin is actuated when the input signal value crosses the threshold of 50% of the nominal input signal, where **Nominal signal value** is a block parameter. The motion is divided into three phases, equal in time: delay (t_1), motion at constant acceleration (t_2), and motion at constant velocity (t_3). The motion stops when the switch-on time (t_{on}) elapses. At this moment, the push-pin reaches the specified stroke value (x_{str}). To return the push-pin into initial position, the control signal must cross back through the threshold of 50% of the nominal input signal, which causes the push-pin to retract. The retract motion follows exactly the same profile but “stretches” over the switch-off time. **Switching-on time** and **Switching-off time** are the block parameters.

The transition in any direction can be interrupted at any time by changing the input signal. If motion is interrupted, the switch-on or switch-off times are proportionally decreased depending on the instantaneous push-pin position.

The push-pin is actuated only by positive signal, similar to the AC or DC electromagnets. The direction of push-pin motion is controlled by the **Actuator orientation** parameter, which can have one of two values: `Acts in positive direction` or `Acts in negative direction`.

Basic Assumptions and Limitations

Push-pin loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.

Parameters

Push-pin stroke

The push-pin stroke. The default value is `0.01` m.

Switching-on time

Time necessary to fully extend the push-pin after the control signal is applied. The default value is `0.1` s.

Switching-off time

Time necessary to retract push-pin from fully extended position after the input signal is removed. The default value is `0.1` s.

Nominal signal value

Sets the value of the nominal input signal. The output motion is initiated as the input signal crosses 50% value of the nominal signal. Other than that, the input signal has no effect on the motion profile. This parameter is meant to reproduce the rated voltage feature of an electromagnet. The default value is `24`.

Initial position

Specifies the initial position of the push-pin. The parameter can have one of two values: `Extended` or `Retracted`. The default value is `Retracted`.

In selecting the initial position, consider the following:

- The steady-state push-pin position always corresponds to the control signal. In other words, zero or negative signal keeps the push-pin at `Retracted`, and positive signal — at `Extended`.

- At the start of simulation, if there is a discrepancy between the initial position of the push-pin, as specified by the **Initial position** parameter, and the control signal, the push-pin immediately starts moving towards the steady-state position designated by the control signal.

Actuator orientation

Parameter controls the direction of the push-pin motion and can have one of two values: `Acts in positive direction` or `Acts in negative direction`. The first value causes the push-pin to move in positive direction, similarly to the action of electromagnet A attached to a directional valve. If the parameter is set to `Acts in negative direction`, the control signal causes the push-pin to move in negative direction from the initial position. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Initial position**
- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has one physical signal input port, associated with the input signal, and one physical signal output port, associated with the output signal (push-pin displacement).

Examples

In the 2-Position Valve Actuator example, the hydraulic circuit contains two actuators. The first one is set to start from the retracted position, while the second one starts from the extended position. Both actuators are driven with a Pulse Generator. The actuators start extending at 1 s, but the second actuator first retracts from 0.01 m to zero, since it was initially extended and there was no signal keeping it there.

In the Single-Acting Cylinder with 3-Way Valve example, the 2-Position Valve Actuator block is used along with a 3-Way Directional Valve block to simulate an electrically operated 3-way directional valve.

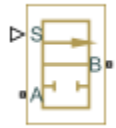
See Also

3-Position Valve Actuator | Hydraulic Double-Acting Valve Actuator | Hydraulic Single-Acting Valve Actuator | Proportional and Servo-Valve Actuator

Introduced in R2006a

2-Way Directional Valve

Hydraulic continuous 2-way directional valve



Library

Directional Valves

Description

The 2-Way Directional Valve block represents a continuous, 2-way directional valve, also referred to as a shut-off valve. It is the device that controls the connection between two lines. The block has two hydraulic connections, corresponding to inlet port (A) and outlet port (B), and one physical signal port connection (S), which controls the spool position. The block is built based on a Variable Orifice block, where the **Orifice orientation** parameter is set to `Opens in positive direction`. This means that positive signal x at port S opens the orifice, and its instantaneous opening h is computed as follows:

$$h = x_0 + x$$

where

h	Orifice opening
x_0	Initial opening
x	Control member displacement from initial position

Because the block is based on a variable orifice, you can choose one of the following model parameterization options:

- `By maximum area and opening` — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.
- `By area vs. opening table` — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement *A=A(h)*.
- `By pressure-flow characteristic` — Use this option if the catalog or data sheet provides a two-dimensional table of the pressure-flow characteristics *q=q(p,h)*.

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table *A=A(h)*. Flow rate is determined analytically, which additionally requires data such as flow discharge coefficient, critical Reynolds number, and fluid density and viscosity. The computation accounts for the laminar and turbulent flow regimes by monitoring the Reynolds number and comparing its value with the critical Reynolds number. See the Variable Orifice block reference page for details. In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause simulation to fail.

In the third case, when an orifice is defined by its pressure-flow characteristics, the flow rate is determined by two-dimensional interpolation. In this case, neither flow regime nor leakage flow rate is taken into account, because these features are assumed to be introduced through the tabulated data. Pressure-flow characteristics are specified with three data sets: array of orifice openings, array of pressure differentials across the orifice, and matrix of flow rate values. Each value of a flow rate corresponds to a specific combination of an opening and pressure differential. In other words, characteristics must be presented as the Cartesian mesh, i.e., the function values must be specified at vertices of a rectangular array. The argument arrays (openings and pressure differentials) must be strictly increasing. The vertices can be nonuniformly spaced. You have a choice of three interpolation methods and two extrapolation methods.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $p = p_A - p_B$. Positive signal at the physical signal port `S` opens the valve.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- Spool loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.

Parameters

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Tabulated valve openings

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly

spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Tabulated valve passage area

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Tabulated flow rates

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is `0`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.

- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

S

Physical signal port to control spool displacement.

Examples

In the Closed-Loop Actuator with 2-Way Valve example, the 2-Way Directional Valve block is used to control the position of a double-acting cylinder. At the start of simulation, the valve is open by 0.42 mm to make the circuit initial position as close as possible to its neutral position.

See Also

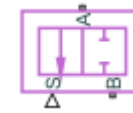
3-Way Directional Valve | 4-Way Directional Valve

Introduced in R2006a

2-Way Directional Valve (G)

Controlled valve with two ports and one flow path

Library: Simscape / Fluids / Gas / Valves & Orifices / Directional Control Valves



Description

The 2-Way Directional Valve (G) block models a valve with two ports (**A** and **B**) and one flow path (**A**–**B**). The path runs through an orifice of variable width, the degree of its opening arising from the motion of a control member. Think of the control member as a spool with a land that covers (by degrees) the **A**–**B** orifice. The distance of the land to the orifice determines if, and to what extent, the orifice is open.

(The distance to the orifice is computed during simulation from the displacement signal specified at port **S**. These variables—in fact, all distances related to spool position—are each defined as a unitless fraction, generally valued between -1 and +1. The calculations are described in detail under “Orifice Opening Fractions” on page 1-15.)

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Valve Positions

The valve is continuously variable. It shifts smoothly between positions, of which it has two: one normal and one working.

The normal position is that to which the valve reverts when it is no longer being operated. The instantaneous displacement of the spool (given at port **S**) is then zero. Unless the land of the spool is installed at an offset to the valve orifice, the valve will be fully closed. The working position is that to which the valve moves when the spool is maximally

displaced (in the positive direction) from the normal position. The orifice is then fully open.

What spool displacement puts the valve in its working position depends on the offset of the (land on) the spool. This offset is generally applied before operation in a real valve, and before simulation in a valve model. Its value is specified in the block as a constant named **Valve opening fraction offset**.

Between valve positions, the opening of the **A**–**B** orifice depends on where, relative to its rim, the land of the spool happens to be. This distance is the orifice opening, and it is normalized here so that its value is a fraction of its maximum (the distance at which is fully open). The normalized variable is referred to as the *orifice opening fraction*.

The orifice opening fraction can range from `0`, in the normal position, to `1`, in the working position. Its value is calculated from the lengths already alluded to: the variable displacement of the control member (applied during operation) and the fixed offset of its land (applied during installation). These give for the orifice opening fraction:

$$h_{\mathbf{AB}} = H_{\mathbf{AB}} + x,$$

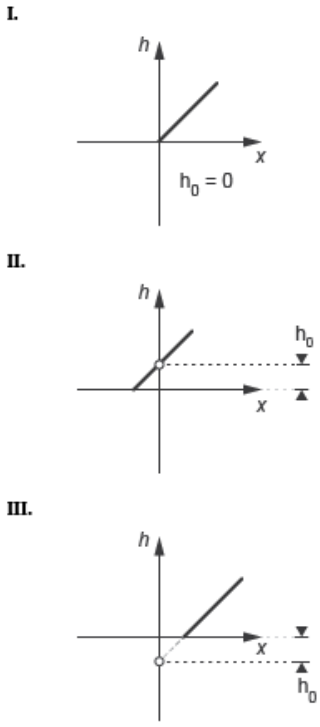
where:

- h is the opening fraction for the sole orifice, **A**–**B**. If the calculation should return a value outside of the range `0`–`1`, the nearest limit is used. (The orifice opening fraction is said to *saturate* at `0` and `1`.)
- H is the opening fraction offset, specified as a block parameter (named **Valve opening fraction offset**). To allow for unusual valve configurations, no limit is imposed on its value, though generally it will fall between `1` and `1`.
- x is the normalized instantaneous displacement of the spool, specified as a physical signal at port **S**. To compensate for equally extreme opening fraction offsets, no limit is imposed on its value (though generally it too will fall near the range of `0` and `1`).

The valve is by default configured so that it is fully closed when its control displacement is zero. Such a valve is often described as being zero-lapped.

It is possible, by offsetting the land of the spool, to model a valve that is underlapped (partially open in the normal valve position) or overlapped (fully closed not only *in* but slightly *beyond* the normal position). The figure shows, for each case, how the orifice opening fraction varies with the spool displacement:

- Case **I**: A zero-lapped valve. The opening fraction offset is zero. When the valve is in the normal position, the land of the spool completely covers the orifice.
- Case **II**: An underlapped valve. The opening fraction offset is positive. When the valve is in the normal position, the land of the spool covers the orifice, but not fully.
- Case **III**: An overlapped valve. The opening fraction offset is negative. The land of the spool completely covers the orifice not only in the normal position but over a small region (of spool displacements) around it.



Control Position in Zero-Lapped, Underlapped, and Overlapped valves

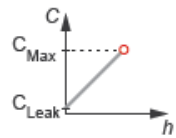
It is common, when picking a valve for throttling or control applications, to match the flow characteristic of the valve to the system it is to regulate.

The flow characteristic relates the opening of the valve to the input that produces it, often spool travel. Here, the opening is expressed as a sonic conductance, flow coefficient, or

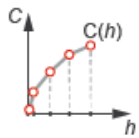
restriction area (the choice between these being given in the **Valve parameterization** setting). The control input is the orifice opening fraction (a function of the spool displacement specified at port **S**).

The flow characteristic is normally given at steady state, with the inlet at a constant, carefully controlled pressure. This (inherent) flow characteristic depends only on the valve and it can be linear or nonlinear, the most common examples of the latter being the quick-opening and equal-percentage types. To capture such flow characteristics, the block provides a choice of opening parameterization (specified in the block parameter of the same name):

- `Linear` — The sonic conductance (C) is a linear function of the orifice opening fraction (h). In the default valve parameterization of `Sonic conductance`, the end points of the line are obtained at opening fractions of `0` and `1` from the **Sonic conductance and leakage flow** and **Sonic conductance at maximum flow** block parameters.



- `Tabulated data` — The sonic conductance is a general function (linear or nonlinear) of the orifice opening fraction. The function is specified in tabulated form, with the columns of the table deriving, in the default valve parameterization, from the **Opening fraction vector** and **Sonic conductance vector** block parameters.



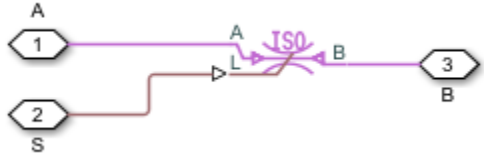
(If the **Valve parameterization** setting is other than `Sonic conductance`, the sonic conductance data is obtained by conversion from the chosen measure of valve opening—such as restriction area or flow coefficient.)

For controlled systems, it is important that the valve, once it is installed, be approximately linear in its flow characteristic. This (installed) characteristic depends on the remainder of the system—it is not generally the same as the inherent characteristic captured in the

block. A pump, for example, may have a nonlinear characteristic that only a nonlinear valve, usually of the equal-percentage type, can adequately compensate for. It is cases of this sort that the `Tabulated data` option primarily targets.

The main purpose of leakage flow is to ensure that no section of a fluid network ever becomes isolated from the rest. Isolated fluid sections can reduce the numerical robustness of the model, slowing down the rate of simulation and, in some cases, causing it to fail altogether. While leakage flow is generally present in real valves, its exact value here is less important than its being a small number greater than zero. The leakage flow area is given in the block parameter of the same name.

This block is a composite component comprising a single Variable Orifice ISO 6358 (G) block connected to ports **A**, **B**, and **S** as shown below. Refer to that block for more detail on the valve parameterizations and block calculations (for example, those used to determine the mass flow rate through the ports).



Ports

Input

S — Valve control signal, `unitless`

physical signal

Instantaneous displacement of the control member against its normal (unactuated) position, specified as a physical signal. The displacement is normalized against the maximum position of the control member (that required to open the orifice fully). See the block description for more information. The signal is unitless and its instantaneous value typically (though not always) in the range of `0`–`1`.

Conserving

`A` — Valve opening
gas

Opening through which flow can enter or exit the valve.

`B` — Valve opening
gas

Opening through which flow can enter or exit the valve.

Parameters

`Valve parameterization` — Method by which to characterize the opening of the valve
`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the `Sonic conductance` parameterization; if a different option is selected, the data specified is converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the calculations of the Variable Orifice ISO 6358 (G) block for detail on the conversions.

`Opening parameterization` — Method by which to calculate the opening characteristics of the valve
`Linear` (default) | `Tabulated data`

Method by which to calculate the opening area of the valve. The default setting treats the opening area as a linear function of the orifice opening fraction. The alternative setting allows for a general, nonlinear relationship to be specified (in tabulated form).

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions
`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value usually reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions

`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive and unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of $\mathtt{ft^3/min}$ (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of `L/min` (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of the leakage value is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause

problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of L/min. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of L/min. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of the leakage value is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Restriction area at maximum flow` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Sum of the gauge pressures at the inlet and pilot port at which the valve is fully open. This value marks the end of the pressure range of the valve (over which the same progressively opens to allow for increased flow).

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Restriction area at leakage flow` — Opening area in the maximally closed position due to sealing imperfections

`1e-10` (default)

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Opening fraction vector` — Orifice opening fractions at which to specify valve opening data

`0 : 0.2 : 1` (default) | unitless vector with elements in the `0`–`1` range

Orifice opening fractions at which to specify the chosen measure of valve opening—sonic conductance, flow coefficient (in SI or USCS forms), or opening area.

This vector must be equal in size to that (or *those*, in the `Sonic conductance` parameterization) containing the valve opening data. The vector elements must be positive and increase monotonically in value from left to right.

Dependencies

This parameter is active and exposed in the block dialog box when the **Opening parameterization** setting is `Tabulated data`.

`Sonic conductance vector` — Vector of sonic conductances at given orifice opening fractions

`[1e-05, .32, .64, .96, 1.28, 1.6] l/s/bar` (default) | vector with units of volume/time/pressure

Sonic conductances at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the orifice opening fraction, sonic conductance, and critical pressure ratio. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of sonic conductance and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Critical pressure ratio vector` — Vector of critical pressure ratios at given orifice opening fractions

`0.3 * ones(1, 6)` (default) | unitless vector

Critical pressure rations at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the orifice opening fraction, sonic conductance, and critical pressure ratio. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of critical pressure ratios and orifice opening fractions—must be of the same size.

The values specified here must each be greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Cv coefficient (USCS) vector` — Vector of flow coefficients, in USCS units, at given orifice opening fractions

`[1e-06, .08, .16, .24, .32, .4]` (default) | vector in units of ft^3/min

Flow coefficients, expressed in US customary units of ft^3/min, at the breakpoints given in the **Opening fraction vector**. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of flow coefficients and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)` and the **Opening parameterization** setting is `Tabulated data`.

`Kv coefficient (SI) vector` — Vector of flow coefficients, in SI units, at given orifice opening fractions

`[1e-06, .06, .12, .18, .24, .3]` (default) | vector in units of L/min

Flow coefficients, expressed in SI units of ft^3/min, at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of flow coefficients and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)` and the **Opening parameterization** setting is `Tabulated data`.

`Restriction area vector` — Vector of opening areas at given orifice opening fractions

`[1e-10, 2e-06, 4e-06, 6e-06, 8e-06, 1e-05] m^2` (default) | vector in units of area

Opening areas at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of opening areas and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area` and the **Opening parameterization** setting is `Tabulated data`.

`Valve opening fraction offset` — Opening fraction of the valve at a zero displacement signal

`0` (default) | unitless scalar

Opening fraction of the valve when the spool displacement is zero. The valve is then in the normal position. The opening fraction measures the distance between the land of the spool and the orifice of the valve, normalized by the maximum allowed.

`Cross-sectional area at ports A and B` — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be of the same size. The flow area specified here should (ideally) match those of the inlets of adjoining components.

`Smoothing factor` — Amount of smoothing to apply to the valve opening function

`0` (default) | positive unitless scalar

Amount of smoothing to apply to the opening function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become. See the Variable Orifice ISO 6358 (G) block for the impact of the smoothing on the block calculations.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings). These can slow down the rate of simulation.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance
`293.15` K (default) | scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance
`1.185` (default) | positive scalar in units of mass/volume

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

See Also

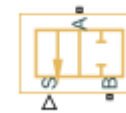
2-Way Directional Valve (G) | 3-Way Directional Valve (G) | 4-Way Directional Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

2-Way Directional Valve (TL)

Valve for modulating flow between two thermal liquid nodes

Library: Simscape / Fluids / Thermal Liquid / Valves & Orifices / Directional Control Valves



Description

The 2-Way Directional Valve (TL) block models the flow through a directional control valve with two ports (**A** and **B**) and one flow path (**A**–**B**). The path contains a variable orifice, which scales in proportion to the displacement of a control member—often a ball, spool, or diaphragm, associated with the signal at port **S**. Valves of this type serve as switches by which to modulate flow in a single line, for example to discharge flow from a tank. The valve model is based on the Variable Area Orifice (TL) block and it shares the equations described for that block.

Valve Positions

The valve is continuously variable. It shifts smoothly between positions, of which it has two. One—the *normal* position—is that to which the valve reverts when its control signal falls to zero. Unless an opening offset has been specified, the **A**–**B** orifice is always fully closed in this position. Another—the *working* position—is that to which the valve moves when its control signal rises to a maximum. The orifice is generally fully open in this position. Note that whether the orifice is in fact open and how open it is both depend on the opening offsets of the valve.

Valve Opening

Which position the valve is in depends on the control member coordinate relative to the **A**–**B** orifice—a length referred to here as the *orifice opening*. The orifice opening is calculated during simulation from the opening offset, specified via the block parameter of the same name, and from the control member displacement, given by the physical signal at port **S**:

$$h_{AB} = h_{AB0} + x,$$

where:

- h_{AB} is the **A**–**B** orifice opening.
- h_{AB0} is the **A**–**B** opening offset.
- x is the control member displacement.

A control member displacement of zero corresponds to a valve that is in its normal position. The **A**–**B** orifice cracks opens when the opening variable (h_{AB}) rises above zero. It then continues to widen with an increasing orifice opening—and therefore with an increasing control member displacement.

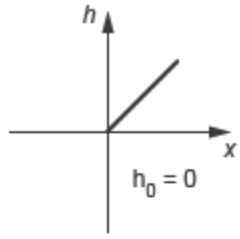
The orifice is fully open when its orifice opening is at a specified maximum. In the linear valve parameterization, this maximum is obtained from the **Maximum valve opening** block parameter. In the tabulated valve parameterizations, the maximum opening is obtained from the last breakpoint in the tabulated data.

Opening Offsets

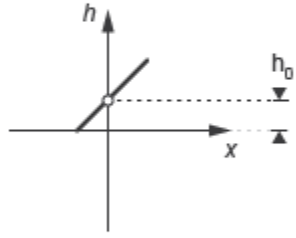
The valve is by default configured so that it is fully closed when the control member displacement is zero. Such a valve is often described as being zero-lapped. It is possible, by applying an offset to the control member, to model a valve that is underlapped (partially open when the control member displacement is zero) or overlapped (fully closed up to a control member displacement equal to the applied offset). The figure shows the orifice opening as a function of the control member displacement for each case:

- Case **I**: A zero-lapped valve. The opening offset is zero. When the valve is in the normal position, the control member completely covers the orifice. The zero-lapped valve is completely closed when the control member displacement falls below zero.
- Case **II**: An underlapped valve. The opening offset is positive. When the valve is in the normal position, the control member covers the orifice but not fully. The underlapped valve is partially open until the control member displacement falls below the negated value of the offset.
- Case **III**: An overlapped valve. The opening offset is negative. The control member completely covers the orifice not only in the normal position but over a small region around it. The overlapped valve is fully closed until the control member crosses the opening offset specified for the orifice.

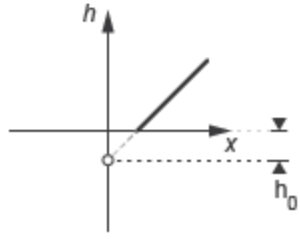
I.



II.



III.

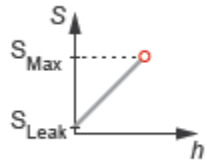


Opening Characteristics

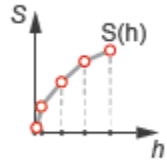
The orifice opening serves during simulation to calculate the mass flow rate through the orifice. The calculation can be a direct mapping from opening to flow rate or an indirect conversion, first from opening to orifice area and then from orifice area to mass flow rate. The calculation, and the data required for it, depend on the setting of the **Valve parameterization** block parameter:

- `Linear area-opening relationship` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve.

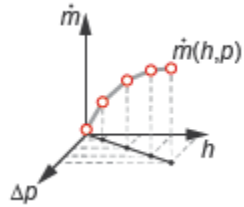
The opening area is assumed to vary linearly with the control member position. The slope of the linear expression is determined from the **Maximum valve opening** and **Maximum opening area** block parameters.



- `Tabulated data - Area vs. opening` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve. The opening area can vary nonlinearly with the control member position. The relationship between the two is given by the tabulated data in the **Valve opening vector** and **Opening area vector** block parameters.



- `Tabulated data - Mass flow rate vs. opening and pressure drop` — Calculate the mass flow rate directly from the control member position and the pressure drop across the valve. The relationship between the three variables can be nonlinear and it is given by the tabulated data in the **Valve opening vector**, **Pressure drop vector**, and **Mass flow rate table** block parameters.



Leakage Flow

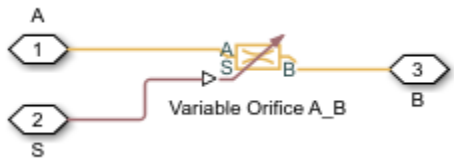
The primary purpose of the leakage flow rate of a closed valve is to ensure that at no time a portion of the thermal liquid network becomes isolated from the remainder of the model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage flow is generally present in real valves but in a model its exact value is less important than its being a small number greater than zero. The leakage flow rate is determined from the **Leakage area** block parameter.

Pressure Loss and Recovery

The pressure drop in the valve is calculated from an empirical parameter known as the discharge coefficient (obtained from the **Discharge coefficient** block parameter). The calculation captures the effect of flow regime, with the pressure drop being proportional to the mass flow rate when the flow is laminar and to the square of the same when the flow is turbulent. Also captured is the pressure recovery than in real valves occurs between the vena contracta (the narrowest aperture of the valve) and the outlet, which generally lies a small distance away.

Composite Component Structure

This block is a composite component comprising a single Variable Area Orifice (TL) block arranged as shown in the figure. The **Orifice orientation** block parameters are set so that a positive signal acts to open the orifice. The specified opening offset is applied to this block. Refer to the Variable Area Orifice (TL) block for detail on the opening area calculations.



Ports

Input

S — Control member displacement, unitless

physical signal

Instantaneous displacement of the valve control member.

Conserving

A — Flow passage

thermal liquid

Opening through which the flow can enter or exit the valve.

B — Flow passage

thermal liquid

Opening through which the flow can enter or exit the valve.

Parameters

Valve parameterization — Method by which to model the opening characteristics of the valve

`Linear area-opening relationship` (default) | `Tabulated data - Area vs. opening` | `Tabulated data - Mass flow rate vs. opening and pressure drop`

Method by which to model the opening characteristics of the valve. The default setting prescribes a linear relationship between the valve opening area and the valve opening. The alternative settings allow for a general, nonlinear relationship to be specified in tabulated form, in one case between the opening area and the control member position (the valve opening), in the other case between the mass flow rate and both the valve opening and the pressure drop between the ports.

Maximum valve opening — Control member position at which the opening area is at a maximum

`5e-3 m` (default) | positive scalar in units of length

Control member position at which the valve is fully open and its opening area is therefore at a maximum. This parameter is used to calculate the slope of the linear expression relating the opening area to the control member position (the valve opening).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Maximum opening area` — Opening area of the valve in the fully open position

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position, when the control member is at the position specified in the **maximum valve opening** block parameter. This parameter is used to calculate the slope of the linear expression relating the opening area to the control member position (the valve opening).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Leakage area` — Opening area of the valve in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Smoothing factor` — Measure of the amount of smoothing to apply to the opening area function

`0.01` (default) | positive unitless scalar

Measure of the amount of smoothing to apply to the opening area function. This parameter determines the widths of the regions to be smoothed, one being at the fully open position, the other at the fully closed position. The smoothing superposes on the linear opening area function two nonlinear segments, one for each region of smoothing.

The greater the value specified, the greater the smoothing and the broader the nonlinear segments.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Valve opening vector` — Vector of control member positions at which to tabulate the valve opening areas

`[-0.002, 0, 0.002, 0.005, 0.015] m` (default) | vector of positive numbers in units of length

Vector of control member positions at which to specify—dependent on the valve parameterization—the opening area of the valve or its mass flow rate. The vector elements must increase monotonically from left to right. This vector must be equal in size to that specified in the **Opening area vector** block parameter or to the number of rows in the **Mass flow rate table** block parameter.

This data serves to construct a one-way lookup table by which to determine, from the control member position, the opening area of the valve or a two-way lookup table by which to determine, from the control member position and pressure drop, the mass flow rate of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Opening area vector` — Vector of opening areas at the specified valve opening breakpoints

`[1e-09, 2.0352e-07, 4.0736e-05, 0.00011438, 0.00034356] m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Valve opening vector** block parameter. The vector elements must increase monotonically from left to right (with increasing control member position). This vector must be equal in size to the number of valve opening breakpoints.

This data serves to construct a one-way lookup table by which to determine from the control member position the opening area of the valve. Data is handled with linear

interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Pressure drop vector` — Vector of pressure drops at which to tabulate the mass flow rates

`[-0.1, 0.3, 0.5, 0.7] MPa` (default) | vector in units of pressure

Vector of pressure differentials from port **A** to port **B** at which to specify the mass flow rate of the valve. The vector elements must increase monotonically from left to right. This vector must be equal in size to the number of columns in the **Mass flow rate table** block parameter.

This data serves to construct a two-way lookup table by which to determine, from the control member position and pressure drop, the opening area of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Mass flow rate table` — Matrix of mass flow rates at the specified valve opening and pressure drop breakpoints

`[-1e-05, 1.7e-05, 2e-05, 2.6e-05; -.002, .0035, .0045, .0053; -.4, .7, .9, 1.06; -1.13, 1.96, 2.5, 3; -3.45, 6, 7.7, 9.13] kg/s` (default) | matrix in units of mass/time

Matrix of mass flow rates corresponding to the breakpoints defined in the **Valve opening vector** and **Pressure drop vector** block parameters. The control member position increases from row to row from top to bottom. The pressure drop increases from column to column from left to right. The mass flow rate must increase monotonically in the same directions (with increasing control member position and increasing pressure drop).

This data serves to construct a two-way lookup table by which to determine, from the control member position and pressure drop, the opening area of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range). Ensure that the number of rows is equal to the

size of the **Opening area vector** block parameter and that the number of columns is equal to the size of the **Pressure drop vector** block parameter.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow temperature` — Inlet temperature at which to specify the tabulated data

`293.15 K` (default) | positive scalar in units of temperature

Nominal inlet temperature, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the temperature measured during simulation.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow pressure` — Inlet pressure at which to specify the tabulated data

`0.101325 MPa` (default) | positive scalar in units of pressure

Nominal inlet pressure, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the pressure measured during simulation.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Valve opening offset` — Distance by which to displace the control member in the normal valve position

`0 m` (default) | positive scalar in units of length

Offset between the control member and the location at which, in the normal orifice position, it would completely cover the orifice. Specify a positive offset to model an underlapped orifice or a negative offset to model an overlapped orifice. For detail on how the opening offsets impact the block calculations, see the block description.

`Cross-sectional area at ports A and B` — Flow area at the thermal liquid ports
`0.01 m` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the valve connects.

`Characteristic longitudinal length` — Measure of the length of the flow path through the valve
`0.1` (default) | positive scalar in units of length

Average distance traversed by the fluid as it travels from inlet to outlet. This distance is used in the calculation of the internal thermal conduction that occurs between the two ports (as part of the smoothed upwind energy scheme employed in the thermal liquid domain).

`Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates
`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes
12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

See Also

3-Way Directional Valve (TL) | 4-Way Directional Valve (TL) | Check Valve (TL) | Variable Area Orifice (TL)

Introduced in R2016a

3-Position Valve Actuator

Actuator for three-position valves

Library

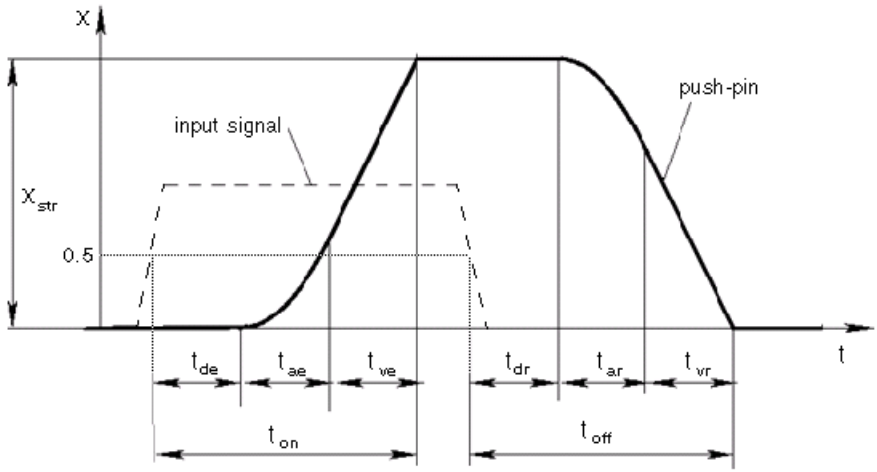
Valve Actuators

Description

The 3-Position Valve Actuator block represents an actuator that you can use with directional valves to control their position. This actuator can drive a three-position valve. The block is developed as a data-sheet-based model and all its parameters are generally provided in catalogs or data sheets. The key parameters are the stroke, switch-on, and switch-off times.

The block has two signal inputs associated with the activation signals for electromagnets A or B. It produces a physical output signal that can be associated with a mechanical translational or rotational push-pin motion. Connect the block output to the directional valve control port.

The actuator is represented as an ideal transducer, where output does not depend on the load exerted on the push-pin and the push-pin motion profile remains the same under any loading conditions. The motion profile represents a typical transition curve for electromagnetic actuators. The following figure shows the motion profile for a case when the input signal is applied long enough for the push-pin to reach the end of the stroke (x_{str}), and then the input signal is removed, causing the push-pin to return to initial position:



The push-pin is actuated when the input signal value crosses the threshold of 50% of the nominal input signal, where **Nominal signal value** is a block parameter. The motion is divided into three phases, equal in time: delay (t_{de}), motion at constant acceleration (t_{ae}), and motion at constant velocity (t_{ve}). The motion stops when the switch-on time (t_{on}) elapses. At this moment, the push-pin reaches the specified stroke value (x_{str}). To return the push-pin into initial position, the control signal must be removed, which causes the push-pin to retract. The retract motion also consists of three phases, equal in time: delay (t_{dr}), motion at constant acceleration (t_{ar}), and motion at constant velocity (t_{vr}). It follows exactly the same profile but “stretches” over the switch-off time. **Switching-on time** and **Switching-off time** are the block parameters.

The signal applied to port A causes the output to move in positive direction. To shift the push-pin in negative direction, you must apply the signal to port B. Only one control signal can be applied at a time. This means that if the actuator is being controlled by the signal at port A, the push-pin must be allowed to return to initial position before the control signal at port B can be processed. The transition in any direction can be interrupted at any time by changing the input signal. If motion is interrupted, the switch-on or switch-off times are proportionally decreased depending on the instantaneous push-pin position.

Only positive signals activate the actuator. In other words, negative signals at ports A and B have no effect on the actuator, which is similar to the behavior of electromagnetically controlled 3-position directional valves.

Basic Assumptions and Limitations

Push-pin loading, such as inertia, spring, hydraulic forces, and so on, is not taken into account.

Parameters

Push-pin stroke

The push-pin stroke. The default value is `0.01` m.

Switching-on time

Time necessary to fully extend the push-pin after the control signal is applied. The default value is `0.1` s.

Switching-off time

Time necessary to retract push-pin from fully extended position after the input signal is removed. The default value is `0.1` s.

Nominal signal value

Sets the value of the nominal input signal. The output motion is initiated as the input signal crosses 50% value of the nominal signal. Other than that, the input signal has no effect on the motion profile. This parameter is meant to reproduce the rated voltage feature of an electromagnet. The default value is `24`.

Initial position

Specifies the initial position of the push-pin. The parameter can have one of three values: `Extended positive`, `Extended negative`, or `Neutral`. The default value is `Neutral`.

In selecting the initial position, consider the following:

- The steady-state push-pin position always corresponds to the control signal. In other words, zero signal keeps the push-pin at `Neutral`, positive signal — at `Extended positive`, and negative signal — at `Extended negative`.
- At the start of simulation, if there is a discrepancy between the initial position of the push-pin, as specified by the **Initial position** parameter, and the control signal, the push-pin immediately starts moving towards the steady-state position designated by the control signal.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Initial position**

All other block parameters are available for modification.

Ports

The block has the following ports:

`A`

Physical signal input port associated with the port A input signal.

`B`

Physical signal input port associated with the port B input signal.

The block also has one physical signal output port, which is associated with the output signal (push-pin displacement).

Examples

In the 3-Position Valve Actuator example, all three actuators are set to different strokes, switch-on and switch-off times, and initial positions. If the initial position is not `Neutral` and the control signal at the beginning of simulation equals zero, the push-pin starts moving towards neutral position, as the actuators A and C show in the example.

See Also

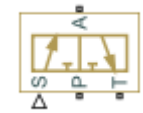
2-Position Valve Actuator | Hydraulic Double-Acting Valve Actuator | Hydraulic Single-Acting Valve Actuator | Proportional and Servo-Valve Actuator

Introduced in R2006a

3-Way Directional Valve

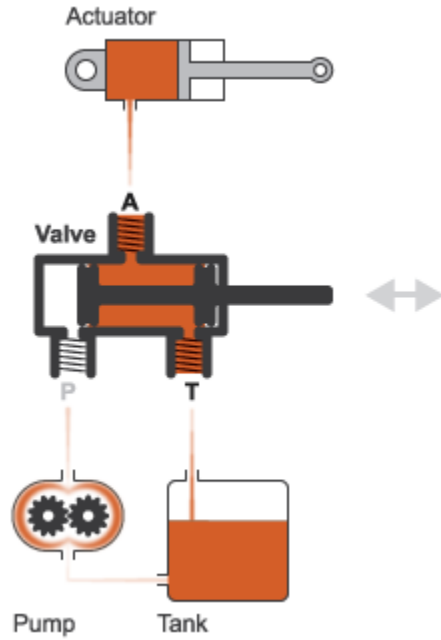
Three-port two-position directional control valve

Library: Simscape / Fluids / Hydraulics (Isothermal) / Valves / Directional Valves



Description

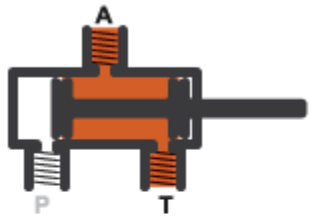
The 3-Way Directional Valve block represents a directional control valve with three ports and two positions, or flow paths. The ports connect to what in a typical model are a hydraulic pump (port **P**), a storage tank (port **T**), and a single-acting actuator (port **A**). Fluid can flow from the pump to the actuator via path **P**-**A** and from the actuator to the tank via path **A**-**T**.



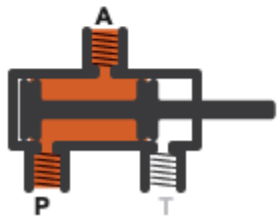
Typical Valve Setup

In the default configuration, one valve position corresponds to the **P-A** flow path maximally—though not necessarily *fully*—open and the **A-T** flow path maximally closed (position **I** in the figure). The second valve position corresponds to the **A-T** flow path maximally open and the **P-A** flow path maximally closed (position **II** in the figure). A translating spool serves as the valve control member and determines the position that the valve is in—**I**, **II**, or in between.

I.



II.



Valve Positions

Physical signal port **S** controls the spool displacement. In the default configuration, a zero displacement corresponds to a fully closed valve between positions **I** and **II**. A positive displacement signal shifts the spool toward valve position **I**. A negative displacement shifts the spool toward valve position **II**. The spool displacement acts indirectly by setting the spool position relative to each flow path—a length known here as the *orifice opening*. The orifice opening in turn determines the opening area of the respective flow path.

Orifice Openings and Offsets

The orifice opening of a flow path depends partly on its *opening offset*—the orifice opening of a flow path at zero spool displacement. The block models only the dynamic effects of the opening offsets. An offset can be due to a change in distance between ports or spool lands—the thick disks built into the spool to obstruct flow. It can also be due to a change in the thicknesses of the spool lands. The orifice openings are computed separately for each flow path in terms of the respective opening offset:

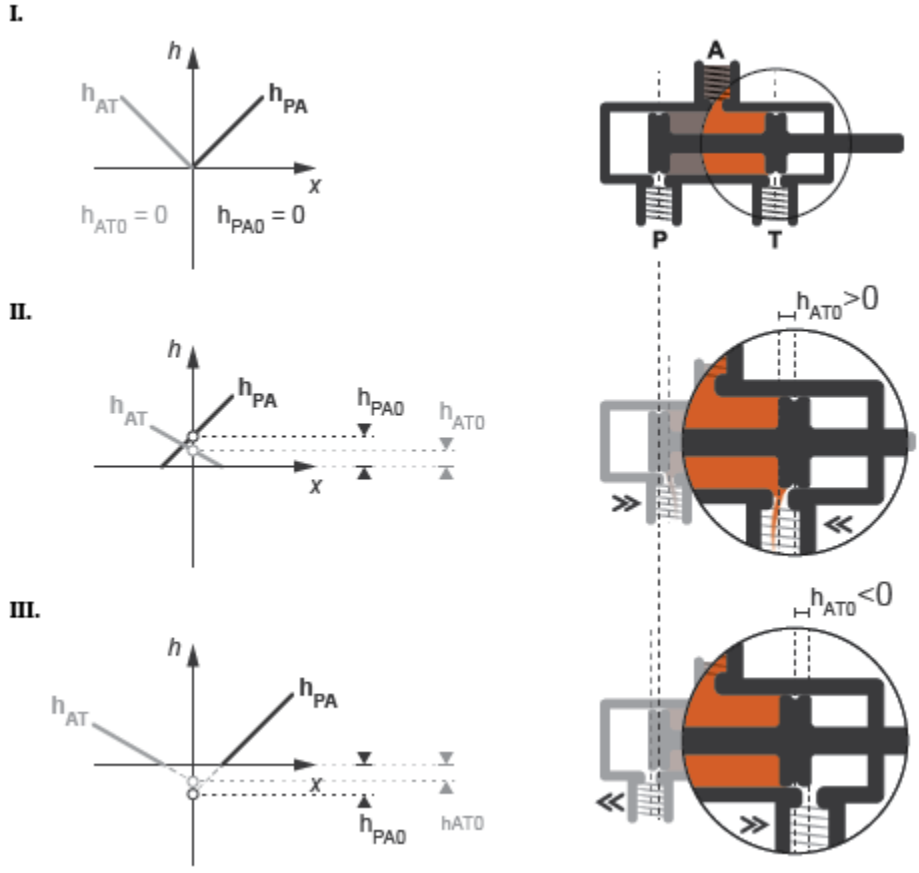
$$h_{PA} = h_{PA0} + x,$$

$$h_{AT} = h_{AT0} - x,$$

where:

- h_{PA} and h_{AT} are the orifice openings of the **P**-**A** and **A**-**T** flow paths. The orifice openings are computed during simulation.
- h_{PA0} and h_{AT0} are the opening offsets of the **P**-**A** and **A**-**T** flow paths. The opening offsets are specified in the **Valve opening offsets** tab.
- x is the spool displacement relative to what in the zero-offset case is a fully closed valve. The spool displacement is specified through physical signal port **S**.

The figure shows the effects of the opening offsets on the orifice openings. Plot **I** corresponds to the default configuration with both opening offsets equal to zero. Plot **II** corresponds to a valve with both opening offsets greater than zero and plot **III** to a valve with both opening offsets smaller than zero. These cases are similar in behavior to zero-lapped (**I**), underlapped (**II**) and overlapped (**III**) valves. The valve schematics to the right show what the offsets might look like for the **A-T** flow path.



Zero (I), Positive (II), and Negative (III) Opening Offsets

An underlapped valve is always partially open and allows some flow at all spool displacements. An overlapped valve is fully closed over an extended range of spool displacements and requires longer spool travel to open. The table summarizes the opening offsets for zero-lapped, underlapped, and overlapped valves. Other configurations are possible—e.g., with one opening offset positive and the other negative.

Valve Lapping	Opening Offsets
Zero-lapped (default)	Both zero

Valve Lapping	Opening Offsets
Underlapped	Both positive
Overlapped	Both negative

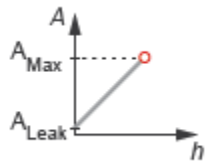
Opening Area Parameterizations

The **Model parameterization** setting determines the calculations used for the opening areas of the flow paths—or, in the `Pressure-flow characteristic` case, the volumetric flow rates. The calculations are based on orifice parameters or tabulated data sets specified in the **Model Parameterization** tab. The block uses the same data for both flow paths if the **Area characteristics** parameter in the **Basic Parameters** tab is set to `Identical for all flow paths` and different data otherwise. Model parameterizations that you can select include:

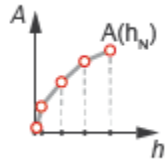
- `Maximum area and opening` — Specify the maximum opening area and the corresponding orifice opening. The opening area is a linear function of the orifice opening,

  $$A_{AT} = \frac{A_{AT,Max}}{h_{AT,Max}} h_{AT} + A_{Leak},$$

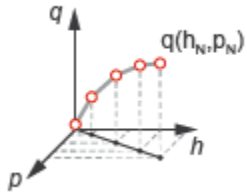
  where A is the opening area and h the orifice opening of a given flow path. The subscript `Max` refers to a fully open orifice and the subscript `Leak` to a fully closed orifice—one with internal leakage flow area only. The figure shows a plot of the linear function $A(h)$.

- `Area vs. opening table` — Specify the opening area at discrete orifice openings as a 1-D lookup table. The opening area is computed for a given orifice opening by interpolation or extrapolation of the tabulated data. The figure shows a conceptual plot of the tabulated function $A(h)$.



- `Pressure-flow characteristic` — Specify the volumetric flow rate at discrete orifice openings and pressure differentials as a 2-D lookup table. The opening area is computed for a given orifice opening and pressure differential by interpolation or extrapolation of the tabulated data. The figure shows a conceptual plot of the tabulated function $q(h, p)$.



Volumetric flow rates are computed analytically in the `Maximum area and opening` and `Area vs. opening table` parameterizations. The calculations are based on additional block parameters such as the flow discharge coefficient and account for the effects of flow regime—laminar or turbulent. Regime transition occurs at the specified critical laminar flow ratio or critical Reynolds number.

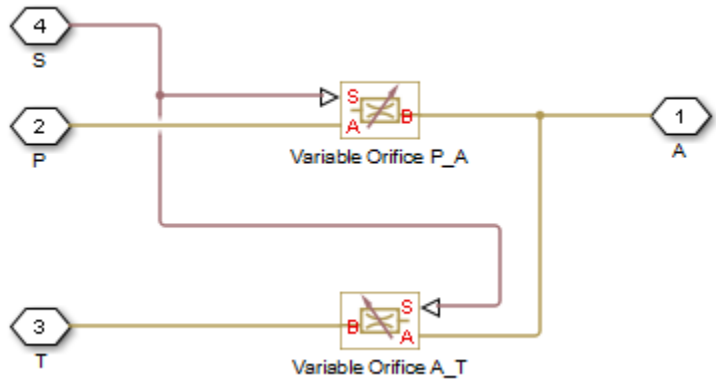
The `Maximum area and opening` and `Area vs. opening table` parameterizations also account for a small leakage area associated with the valve spool-to-bore clearance. The leakage area ensures that portions of the hydraulic network never become isolated when a flow path is closed. Isolated, or “hanging”, network portions affect computational efficiency and can cause simulation to fail.

The effects of flow regime and internal leakage are assumed to be reflected in the tabulated flow rate data specified directly in the `Pressure-flow characteristic` parameterization.

Structural Component Diagram

The block is a composite component with two Variable Orifice blocks driven by a single physical signal. Block `Variable Orifice P_A` represents the **P-A** flow path. Block `Variable orifice A_T` represents the **A-T** flow path. The physical signal is specified through Connection Port block **S**.

The **Orifice orientation** block parameters are set so that a positive signal acts to open `Variable Orifice P_A` while closing `Variable Orifice A_T`. A negative signal has the opposite effect, acting to open `Variable Orifice A_T` while closing `Variable Orifice P_A`.



Valve Structural Diagram

Assumptions

- Fluid inertia is ignored.
- Spool loading due to inertial, spring, and other forces is ignored.
- All valve orifices are assumed identical in size unless otherwise specified.

Ports

Input

S — Spool displacement, m

physical signal

Physical signal input port for the spool displacement.

Conserving

P — Supply line orifice

hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) port associated with the supply line orifice.

T — Return line orifice

hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) port associated with the return line orifice.

A — Actuation line orifice

hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) port associated with the actuation orifice.

Parameters

Basic Parameters

Area characteristics — Symmetry of the flow path opening characteristics

`Identical for all flow paths` (default) | `Different for each flow path`

Choice of different or identical flow path opening characteristics. Select `Different for each flow path` to specify flow path parameters or tabulated data separately for each flow path.

Model parameterization — Types of valve parameters to use as input

`Maximum area and opening` (default) | `Area vs. opening table` | `Pressure-flow characteristic`

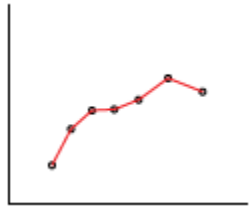
Parameterization of the valve model. Options include:

- `By maximum area and opening` — Specify the maximum orifice opening and opening area. The opening area varies linearly with the spool displacement specified at physical signal port **S**.
- `By area vs. opening table` — Specify the flow path opening area at discrete orifice openings as a 1-D lookup table. The opening area is computed by interpolation or extrapolation of the tabulated data.
- `By pressure-flow characteristic` — Specify the flow path volumetric flow rates at discrete orifice openings and pressure differentials. The flow rate is computed by interpolation or extrapolation of the tabulated data.

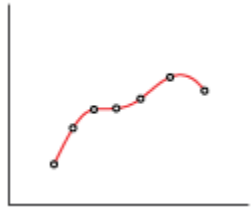
`Interpolation method` — Method of computing values inside the lookup table range

`Linear` (default) | `Smooth`

Method of computing values inside the tabulated data range. The `Linear` method joins adjacent data points with straight line or surface segments with generally discontinuous slope at the segment boundaries. Surface segments are used in the 2-D lookup table specified in the `Pressure-flow characteristic` model parameterization.



The `Smooth` method replaces the straight segments with curved versions that have continuous slope everywhere inside the tabulated data range. The segments form a smooth line or surface passing through all of the tabulated data points without the discontinuities in first-order derivatives characteristic of the `Linear` interpolation method.



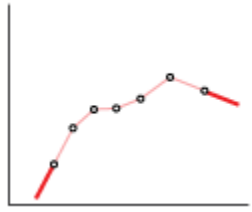
Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By area vs. opening table` or `Pressure-flow characteristic`.

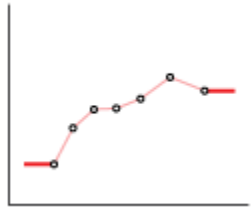
`Extrapolation method` — Method of computing values outside of the lookup table range

`Linear` (default) | `Nearest`

Method of computing values outside of the tabulated data range. The `Linear` method extends the line segment drawn between the last two data points at each end of the data range outward with a constant slope.



The `Nearest` method extends the last data point at each end of the data range outward as a horizontal line with constant value.



Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By area vs. opening table` or `Pressure-flow characteristic`.

`Flow discharge coefficient` — Semi-empirical parameter for valve capacity characterization

`0.7` (default) | unitless scalar between 0 and 1

Ratio of the actual and theoretical flow rates through the valve. This parameter depends on the geometrical properties of the valve. Values are usually provided in textbooks and manufacturer data sheets.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table`.

`Leakage area` — Internal leakage area between valve inlets

`1e-12 m^2` (default) | positive scalar in units of area

Total area of internal leaks in the completely closed state. The purpose of this parameter is to maintain the numerical integrity of the fluid network by preventing a portion of that network from becoming isolated when the valve is completely closed.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table`.

`Laminar transition specification` — Choice of parameter used to define the laminar-turbulent transition

`Pressure ratio` (default) | `Reynolds number`

Select the parameter to base the laminar-turbulent transition on. Options include:

- `Pressure ratio` — Flow transitions between laminar and turbulent at the pressure ratio specified in the **Laminar flow pressure ratio** parameter. Use this option for the smoothest and most numerically robust flow transitions.
- `Reynolds number` — The transition occurs at the Reynolds number specified in the **Critical Reynolds number** parameter. Flow transitions are more abrupt and can cause simulation issues at near-zero flow rates.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table`.

`Laminar flow pressure ratio` — Pressure ratio at the laminar-turbulent flow transition

`0.999` (default) | unitless scalar between 0 and 1

Pressure ratio at which the flow transitions between the laminar and turbulent regimes. The pressure ratio is the fraction of the outlet pressure over the inlet pressure.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table` and the **Laminar transition specification** parameter is set to `Reynolds number`.

`Critical Reynolds number` — Reynolds number at the laminar-turbulent flow transition

12 (default) | unitless positive scalar

Maximum Reynolds number for laminar flow. This parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value, 12, corresponds to a round orifice in thin material with sharp edges.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table` and the **Laminar transition specification** parameter is set to `Reynolds number`.

Model Parameterization: Maximum area and opening

`Maximum opening` — Orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Orifice opening at which the opening area of a flow path is a maximum. The maximum opening is the same for all flow paths when this parameter is exposed. For the orifice opening definitions, see “Orifice Openings and Offsets” on page 1-45.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`Maximum opening area` — Maximum cross-sectional area of a flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of a flow path. The maximum opening areas are the same for all paths when this parameter is exposed. For the opening area calculations, see “Opening Area Parameterizations” on page 1-48.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`P-A, maximum opening` — `P-A` orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Spool position relative to the **P-A** flow path at which the **P-A** opening area is equal to the **P-A, maximum opening area** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`P-A, maximum opening area` — Maximum flow area of the `P-A` flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of the **P-A** flow path. The maximum opening areas are different for each flow path when this parameter is exposed. For the opening area calculations, see “Opening Area Parameterizations” on page 1-48.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`A-T, maximum opening` — `A-T` Orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Spool position relative to the **A-T** flow path at which the **A-T** opening area is equal to the **A-T, maximum opening area** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`A-T, maximum opening area` — Maximum flow area of the `A-T` flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of the **P-A** flow path. The maximum opening areas are different for each flow path when this parameter is exposed. For the opening area calculations, see “Opening Area Parameterizations” on page 1-48.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

Model Parameterization: Area vs. opening table

`Opening vector` — Orifice openings at which to specify the flow path opening areas

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of orifice openings used to construct the opening area 1-D lookup table for all flow paths. The opening characteristics are identical for all flow paths when this parameter is exposed. The number of elements in the array determines the number of columns in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`Opening area vector` — Opening areas corresponding to the orifice openings specified in the `Opening vector` parameter

`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of flow path opening areas corresponding to the specified orifice openings. The number of elements in the array must match the number of elements in the **Opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`P-A, opening vector` — `P-A` orifice openings at which to specify the `P-A` opening area

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **P-A** orifice openings used to construct the opening area 1-D lookup table for the **P-A** flow path. The number of elements in the array determines the number of elements in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`P-A, opening area vector` — `P-A` opening areas corresponding to the orifice openings specified in the `P-A, opening vector` parameter
`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of **P-A** opening areas corresponding to the specified **P-A** orifice openings. The number of elements in the array must match the number of elements in the **P-A, opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`A-T, opening vector` — `A-T` orifice openings at which to specify the `A-T` opening area
`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **A-T** orifice openings used to construct the opening area 1-D lookup table for the **A-T** flow path. The number of elements in the array determines the number of elements in the table.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`A-T, opening area vector` — `A-T` opening areas corresponding to the orifice openings specified in the `A-T, opening vector` parameter
`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of **A-T** opening areas corresponding to the specified **A-T** orifice openings. The number of elements in the array must match the number of elements in the **A-T, opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

Model Parameterization: Pressure-flow characteristic

`Opening vector, s` — Orifice openings at which to specify the flow rate tabulated data

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of orifice openings used to construct the volumetric flow rate 2-D lookup table. The number of elements in the array determines the number of rows in the table. The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`Pressure differential vector, dp` — Pressure differentials at which to specify the flow rate data

`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table. The number of elements in the array determines the number of columns in the table. The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`Volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified orifice openings and pressure differentials

5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates corresponding to the specified orifice openings and pressure differentials. The number of rows must match the number of elements in the

Opening vector, s parameter. The number of columns must match the number of elements in the **Pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-A, opening vector, s` — Orifice openings at which to specify the `P-A` flow rate data

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **P-A** orifice openings used to construct the **P-A** volumetric flow rate 2-D lookup table. The number of elements in the array determines the number of rows in the table. The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-A, pressure differential vector, dp` — Pressure differentials at which to specify the `P-A` flow rate data

`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table for the **P-A** flow path. The number of elements in the array determines the number of columns in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-A, volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified `P-A` orifice openings and pressure differentials
5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates through the **P-A** flow path at the **P-A** orifice openings and pressure differentials. The number of rows must match the number of elements in the **P-A, opening vector, s** parameter. The number of columns must match the number of elements in the **P-A, pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`A-T, opening vector, s` — Orifice openings at which to specify the `A-T` flow rate data
`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **A-T** orifice openings used to construct the **A-T** volumetric flow rate 2-D lookup table. The number of elements in the array determines the number of rows in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`A-T, pressure differential vector, dp` — Pressure differentials at which to specify the `A-T` flow rate data
`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table for the **A-T** flow path. The number of elements in the array determines the number of columns in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`A-T, volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified `A-T` orifice openings and pressure differentials

5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates through the **A-T** flow path at the **A-T** orifice openings and pressure differentials. The number of rows must match the number of elements in the **A-T, opening vector, s** parameter. The number of columns must match the number of elements in the **A-T, pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

Valve Opening Offsets

`Between P-A` — `P-A` orifice opening at zero spool displacement

`0 m` (default) | scalar number with units of length

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve.

`Between A-T` — `A-T` orifice opening at zero spool displacement

`0 m` (default) | scalar number with units of length

Orifice opening of the **A-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve.

See Also

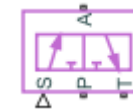
2-Way Directional Valve | 4-Way Directional Valve

Introduced in R2017a

3-Way Directional Valve (G)

Controlled valve with three ports and two flow paths

Library: Simscape / Fluids / Gas / Valves & Orifices / Directional Control Valves



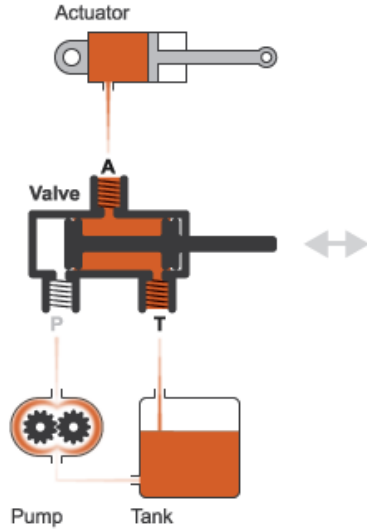
Description

The 3-Way Directional Valve (G) block models a valve with three gas ports (**P**, **A**, and **T**) and two flow paths to switch between (**P**–**A** and **A**–**T**). The paths each run through an orifice of variable width, its opening here tied to the motion of a control member. Think of the control member as a spool with two lands to cover (by degrees) the **P**–**A** and **A**–**T** orifices. The distance of a land to its appointed orifice determines if, and to what extent, that orifice is open.

(The distances from the lands to the orifices are computed during simulation from the displacement signal specified at port **S**. They, and all distances related to spool position, are defined as unitless fractions, generally valued between -1 and +1. The calculations are described in detail under “Orifice Opening Fractions” on page 1-67.)

The gas ports connect to what, in a representative system, are a pump (**P**), a tank (**T**), and a single-acting actuator (**A**). Opening the **P**–**A** flow path (while closing **A**–**T**) allows the pump to pressurize the actuator. The shaft of the latter translates (normally to extend). Opening the alternate (**A**–**T**) flow path (while closing **P**–**A**) allows the tank to absorb the excess pressure from the actuator, freeing the shaft to translate in reverse (for example, by the weight of the translated load).

The port connections will vary with the system being modeled, but the purpose of the valve—to switch between flow paths and to regulate the flow passing through them—should not.



A Common Use of a 3-Way Directional Valve

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

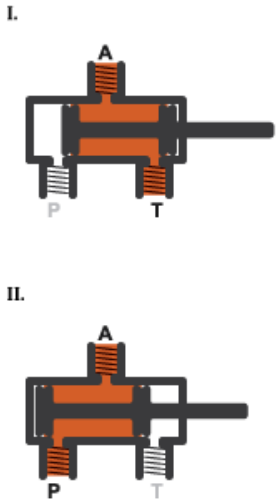
Valve Positions

The valve is continuously variable. It shifts smoothly between positions, of which it has three: one normal and two working.

The normal position is that to which the valve reverts when it is no longer being operated. The instantaneous displacement of the spool (given at port **S**) is then zero. Unless the lands of the spool are installed at an offset to their orifices, the valve will be fully closed.

The working positions are those to which the valve moves when the spool is maximally displaced, in either positive or negative direction, from the normal position. One orifice will then generally be closed and the other open to full capacity. If the displacement is in the negative direction, the **P**–**A** orifice is closed and the **A**–**T** orifice open (position **I** in the

figure). If the displacement is in the positive direction, the **P**–**A** orifice is open and the **A**–**T** orifice closed (position **II**).



What spool displacement puts the valve in a working position depends on the offsets of the lands on the spool. These are generally applied before operation, in a real valve, and before simulation, in a valve model. They are specified in the block as constants (fixed from the start of simulation) in the **Valve Opening Fraction Offsets** tab.

Between valve positions, the opening of an orifice depends on where, relative to its rim, its land of the spool happens to be. This distance is the orifice opening, and it is normalized here so that its value is a fraction of its maximum (the distance at which the orifice is fully open). The normalized variable is referred to here as the *orifice opening fraction*.

The orifice opening fractions range from -1 in working position **I** to +1 in working position **II** (using the labels shown in the figure).

The opening fractions are calculated from the lengths already alluded to: the variable displacement of the control member (applied during operation) and the fixed offsets of its lands (applied during installation). These lengths are themselves defined as unitless fractions of the maximum land–orifice distance. (The offsets are referred to here as the *opening fraction offsets*.)

The opening fraction of the **P**–**A** orifice is:

$$h_{\mathbf{PA}} = H_{\mathbf{PA}} + x.$$

That of the **A**–**T** orifice is:

$$h_{\mathbf{AT}} = H_{\mathbf{AT}} - x.$$

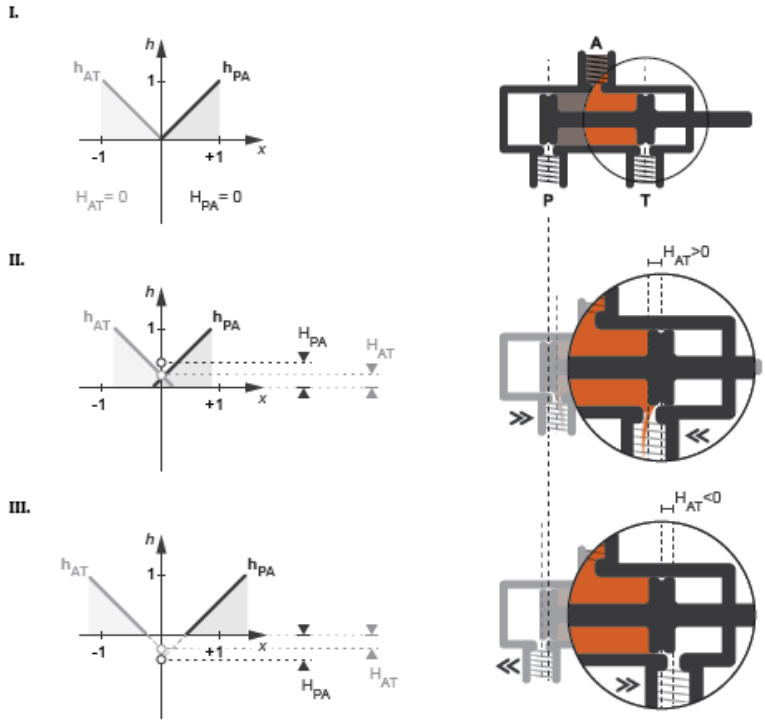
In both equations:

- *h* is the opening fraction of the orifice denoted by the subscript (**P**–**A** or **A**–**T**). If the calculation should return a value outside of the range **0**–**1**, the nearest limit is used. (The orifice opening fractions are said to *saturate* at `0` and `1`.)
- *H* is the opening fraction offset for the orifice denoted by the subscript. The offsets are each specified as a block parameter (in the **Valve Opening Fraction Offsets** tab). To allow for unusual valve configurations, no limit is imposed on their values, though generally these will fall between `-1` and `+1`.
- *x* is the normalized instantaneous displacement of the spool, specified as a physical signal at port **S**. To compensate for equally extreme opening fraction offsets, no limit is imposed on its value (though it generally will fall near the range of `-1` to `+1`.)

The valve is by default configured so that it is fully closed when the spool displacement is zero. Such a valve is often described as being *zero-lapped*.

It is possible, by offsetting the lands of the spool, to model a valve that is underlapped (partially open in the normal valve position) or overlapped (fully closed not only *in* but also slightly *beyond* the normal position). The figure shows, for each case, how the orifice opening fractions vary with the instantaneous spool displacement:

- Case **I**: A zero-lapped valve. The opening fraction offsets are both zero. When the valve is in the normal position, the lands of the spool completely cover both orifices.
- Case **II**: An underlapped valve. The opening fraction offsets are both positive. When the valve is in the normal position, the lands of the spool cover both orifices, but neither fully.
- Case **III**: An overlapped valve. The opening fraction offsets are both negative. The lands of the spool completely cover both orifices not only in the normal position but over a small region (of spool displacements) around it.



Opening Characteristics

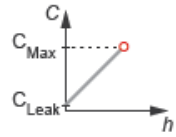
It is common, when picking a valve for throttling or control applications, to match the flow characteristic of the valve to the system it is to regulate.

The flow characteristic relates the opening of the valve to the input that produces it, often spool travel. Here, the opening is expressed as a sonic conductance, flow coefficient, or restriction area (the choice between them being given by the **Valve parameterization** setting). The control input is the orifice opening fraction (a function of the spool displacement specified at port **S**).

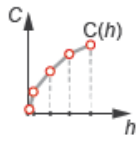
The flow characteristic is normally given at steady state, with the inlet at a constant, carefully controlled pressure. This (inherent) flow characteristic depends only on the valve and it can be linear or nonlinear, the most common examples of the latter being the quick-opening and equal-percentage types. To capture such flow characteristics, the block

provides a choice of opening parameterization (specified in the block parameter of the same name):

- `Linear` — The sonic conductance (C) is a linear function of the orifice opening fraction (h). In the default valve parameterization of `Sonic conductance`, the end points of the line are obtained at opening fractions of `0` and `1` from the **Sonic conductance and leakage flow** and **Sonic conductance at maximum flow** block parameters.



- `Tabulated data` — The sonic conductance is a general function (linear or nonlinear) of the orifice opening fraction. The function is specified in tabulated form, with the columns of the table deriving, in the default valve parameterization, from the **Opening fraction vector** and **Sonic conductance vector** block parameters.



(If the **Valve parameterization** setting is other than `Sonic conductance`, the sonic conductance data is obtained by conversion from the chosen measure of valve opening (such as restriction area or flow coefficient). The opening data applies to both orifices equally.)

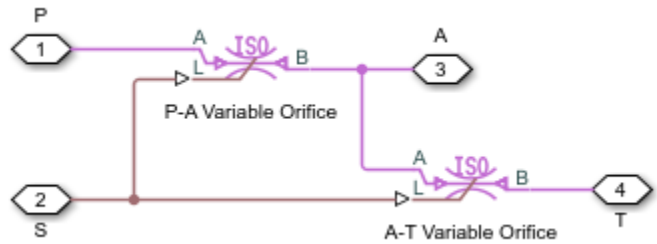
For controlled systems, it is important that the valve, once it is installed, be approximately linear in its flow characteristic. This (installed) characteristic depends on the remainder of the system—it is not generally the same as the inherent characteristic captured in the block. A pump, for example, may have a nonlinear characteristic that only a nonlinear valve, usually of the equal-percentage type, can adequately compensate for. It is cases of this sort that the `Tabulated data` option primarily targets.

Leakage Flow

The main purpose of leakage flow is to ensure that no section of a fluid network ever becomes isolated from the rest. Isolated fluid sections can reduce the numerical robustness of the model, slowing down the rate of simulation and, in some cases, causing it to fail altogether. While leakage flow is generally present in real valves, its exact value here is less important than its being a small number greater than zero. The leakage flow area is given in the block parameter of the same name.

Composite Structure

This block is a composite component comprising two instances of the Variable Orifice ISO 6358 (G) block connected to ports **P**, **A**, **T**, and **S** as shown below. Refer to that block for more detail on the valve parameterizations and block calculations (for example, those used to determine the mass flow rate through the ports).



Ports

Input

S — Valve control signal, `unitless`

physical signal

Instantaneous displacement of the control member against its normal (unactuated) position, specified as a physical signal. The displacement is normalized against the maximum position of the control member (that required to open the orifice fully). See the block description for more information. The signal is unitless and its instantaneous value typically (though not always) in the range of -1–+1.

Conserving

A — Valve opening
gas

Opening through which flow can enter or exit the valve.

B — Valve opening
gas

Opening through which flow can enter or exit the valve.

T — Valve opening
gas

Opening through which flow can enter or exit the valve.

Parameters

Basic Parameters

`Valve parameterization` — Method by which to characterize the opening of the valve
`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the `Sonic conductance` parameterization; if a different option is selected, the data specified is converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the calculations of the Variable Orifice ISO 6358 (G) block for detail on the conversions.

`Opening parameterization` — Method by which to calculate the opening characteristics of the valve
`Linear` (default) | `Tabulated data`

Method by which to calculate the opening area of the valve. The default setting treats the opening area as a linear function of the orifice opening fraction. The alternative setting allows for a general, nonlinear relationship to be specified (in tabulated form).

`Cross-sectional area at ports A, B, and T` — Area normal to the flow path at the valve ports
`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be of the same size. The flow area specified here should (ideally) match those of the inlets of adjoining components.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes
`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance
`293.15 K` (default) | scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance
`1.185` (default) | positive scalar in units of mass/volume

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

Model Parameterization

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions
`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value usually reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions

`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive and unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of $\mathtt{ft^3/min}$ (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of $\mathtt{ft^3/min}$ (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of the leakage value is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause

problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of L/min. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of L/min. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of the leakage value is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Restriction area at maximum flow` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Sum of the gauge pressures at the inlet and pilot port at which the valve is fully open. This value marks the end of the pressure range of the valve (over which the same progressively opens to allow for increased flow).

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Restriction area at leakage flow` — Opening area in the maximally closed position due to sealing imperfections

`1e-10` (default)

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Opening fraction vector` — Orifice opening fractions at which to specify valve opening data

`0 : 0.2 : 1` (default) | unitless vector with elements in the `0`–`1` range

Orifice opening fractions at which to specify the chosen measure of valve opening—sonic conductance, flow coefficient (in SI or USCS forms), or opening area.

This vector must be equal in size to that (or *those*, in the `Sonic conductance` parameterization) containing the valve opening data. The vector elements must be positive and increase monotonically in value from left to right.

Dependencies

This parameter is active and exposed in the block dialog box when the **Opening parameterization** setting is `Tabulated data`.

`Sonic conductance vector` — Vector of sonic conductances at given orifice opening fractions

`[1e-05, .32, .64, .96, 1.28, 1.6] l/s/bar` (default) | vector with units of volume/time/pressure

Sonic conductances at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the orifice opening fraction, sonic conductance, and critical pressure ratio. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of sonic conductance and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Critical pressure ratio vector` — Vector of critical pressure ratios at given orifice opening fractions

`0.3 * ones(1, 6)` (default) | unitless vector

Critical pressure rations at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the orifice opening fraction, sonic conductance, and critical pressure ratio. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of critical pressure ratios and orifice opening fractions—must be of the same size.

The values specified here must each be greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Cv coefficient (USCS) vector` — Vector of flow coefficients, in USCS units, at given orifice opening fractions

`[1e-06, .08, .16, .24, .32, .4]` (default) | vector in units of ft^3/min

Flow coefficients, expressed in US customary units of ft^3/min, at the breakpoints given in the **Opening fraction vector**. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of flow coefficients and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)` and the **Opening parameterization** setting is `Tabulated data`.

`Kv coefficient (SI) vector` — Vector of flow coefficients, in SI units, at given orifice opening fractions

`[1e-06, .06, .12, .18, .24, .3]` (default) | vector in units of L/min

Flow coefficients, expressed in SI units of `L/min`, at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of flow coefficients and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)` and the **Opening parameterization** setting is `Tabulated data`.

`Restriction area vector` — Vector of opening areas at given orifice opening fractions

`[1e-10, 2e-06, 4e-06, 6e-06, 8e-06, 1e-05] m^2` (default) | vector in units of area

Opening areas at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of opening areas and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area` and the **Opening parameterization** setting is `Tabulated data`.

`Smoothing factor` — Amount of smoothing to apply to the valve opening function

`0` (default) | positive unitless scalar

Amount of smoothing to apply to the opening function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become. See the Variable Orifice ISO 6358 (G) block for the impact of the smoothing on the block calculations.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings). These can slow down the rate of simulation.

Valve Opening Fraction Offsets

`Between Ports P and A` — Opening fraction offset for the P–A orifice
`0` (default) | unitless scalar

Opening fraction of the **P**–**A** orifice when the spool displacement is zero. The valve is then in the normal position. The opening fraction measures the distance of a land of the spool to its appointed orifice (here **P**–**A**), normalized by the maximum such distance. It is unitless and (generally) between `0` and `1`.

`Between Ports A and T` — Opening fraction offset for the A–T orifice
`0` (default) | unitless scalar

Opening fraction of the **A**–**T** orifice when the spool displacement is zero. The valve is then in the normal position. The opening fraction measures the distance of a land of the spool to its appointed orifice (here **A**–**T**), normalized by the maximum such distance. It is unitless and (generally) between `0` and `1`.

See Also

2-Way Directional Valve (G) | 4-Way Directional Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

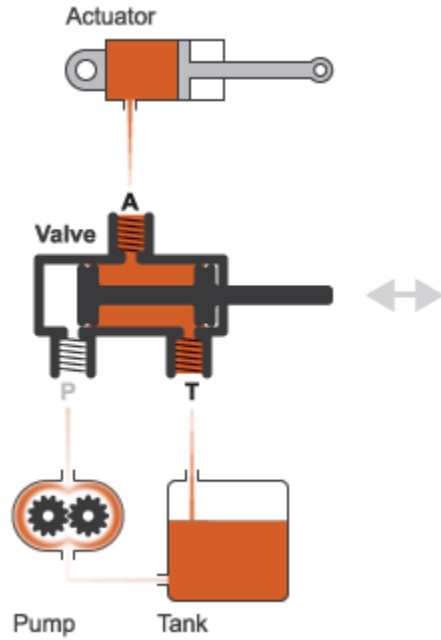
3-Way Directional Valve (TL)

Valve for routing flow at the junction of three lines

Library: Simscape / Fluids / Thermal Liquid / Valves & Orifices / Directional Control Valves

Description

The 3-Way Directional Valve block models the flow through a directional control valve with three ports (**P**, **T**, and **A**) and two flow paths (the lines **P**-**A** and **A**-**T**). The ports connect to what in a typical system are a pump (**P**), a tank (**T**), and a single-acting actuator (**A**). The paths each contain a variable orifice that opens in proportion to the displacement of a control member—often a ball, spool, or diaphragm, associated with the signal at port **S**. This valve serves as a switch by which to partition the flow between the branches of a three-pronged junction.



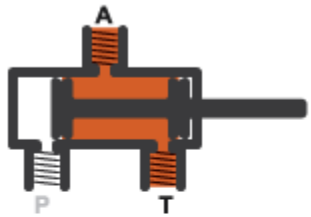
Typical Valve Setup

Valve Positions

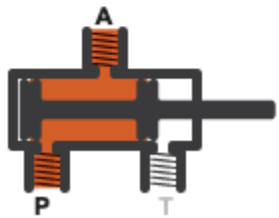
The valve is continuously variable, which is to say that it shifts smoothly between positions. Of these there are three: one normal and two working. The normal position is that to which the valve reverts when the control member is no longer being actuated. The orifices are both typically closed in this position. The working positions are those to which the valve moves when the control member is maximally actuated. One orifice is typically closed in this position while the other is fully open. The exact opening states of the orifices depend on the opening offsets specified in the block dialog box.

The working positions are shown in the figure in the default case of a valve without opening offsets. One, labeled **I**, corresponds to the **A**-**T** orifice being maximally open and the **P**-**A** orifice maximally closed. The other, labeled **II**, corresponds to the reverse arrangement, with the **P**-**A** orifice being maximally open and the **A**-**T** orifice maximally closed. At points between the normal and working positions, one orifice is partially open while the other is fully closed. Note that no physical connection exists between ports **P** and **T** and therefore that no flow can develop across them.

I.



II.



Valve Opening

Which position the valve is in depends on the control member coordinates relative to the **P**-**A** and **A**-**T** orifices—lengths referred to here as the *orifice openings*. These are calculated during simulation from their opening offsets, each specified as a block parameter in the block dialog box, and from the control member displacement, given by the physical signal at port **S**. For the **P**-**A** orifice:

$$h_{PA} = h_{PA0} + x,$$

where:

- h_{PA} is the **P**-**A** orifice opening.
- h_{PA0} is the **P**-**A** opening offset.
- x is the control member displacement. A control member displacement of zero corresponds to a valve that is in the normal state.

For the **A**-**T** orifice:

$$h_{AT} = h_{AT0} + x,$$

where:

- h_{AT} is the **A**-**T** orifice opening.
- h_{AT0} is the **A**-**T** opening offset.

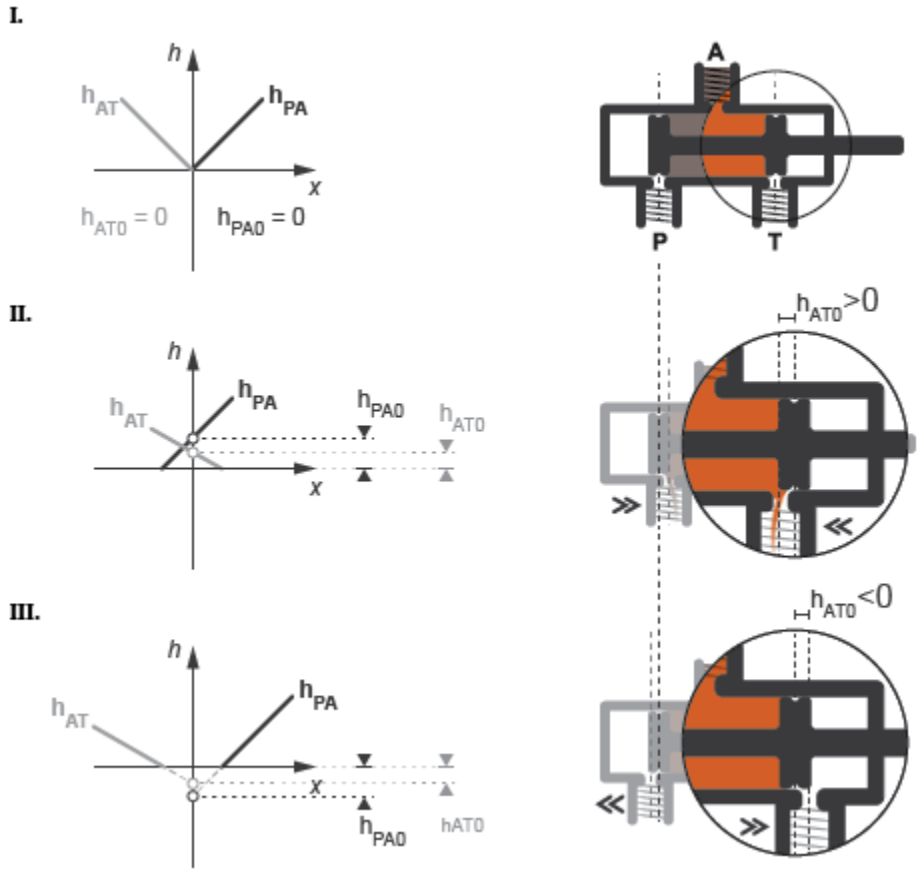
An orifice cracks opens when its calculated opening (variable h) rises above zero. It then continues to widen with a rising opening value. In the case of the **P**-**A** orifice, this occurs as the control member is displaced in the positive direction. In the case of the **A**-**T** orifice, it occurs as the control member is displaced in the negative direction.

The orifices are each fully open when the opening value is at a specified maximum. In the linear valve parameterization, this maximum is obtained from the **Maximum valve opening** block parameter. You can set the **Area characteristics** parameter to `Different for each flow path` to specify the **Maximum valve opening** parameter separately for each orifice. In tabulated valve parameterizations, the maximum opening is obtained from the last breakpoint in the tabulated data.

Opening Offsets

The valve is by default configured so that it is fully closed when the control member displacement is zero. Such a valve is often described as being zero-lapped. It is possible, by applying an offset to the control member, to model a valve that is underlapped (partially open when the control member displacement is zero) or overlapped (fully closed up to a control member displacement equal to the applied offset). The figure shows the orifice opening as a function of the control member displacement for each case:

- Case **I**: A zero-lapped valve. The opening offsets are both zero. When the valve is in the normal position, the control member completely covers both orifices. The zero-lapped valve is completely closed when the control member displacement is exactly zero.
- Case **II**: An underlapped valve. The opening offsets are both positive. When the valve is in the normal position, the control member covers both orifices but neither fully. The underlapped valve is always at least partially open.
- Case **III**: An overlapped valve. The opening offsets are both negative. The control member completely covers both orifices not only in the normal position but over a small region around it. The overlapped valve is fully closed until the control member crosses the opening offset of either orifice.

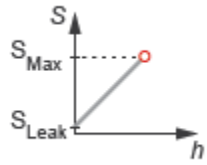


Opening Characteristics

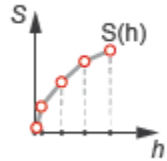
The orifice openings serve during simulation to calculate the mass flow rates through the orifices. The calculation can be a direct mapping from opening to flow rate or an indirect conversion, first from opening to orifice area and then from orifice area to mass flow rate. The calculation, and the data required for it, depend on the setting of the **Valve parameterization** block parameter:

- `Linear area-opening relationship` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve.

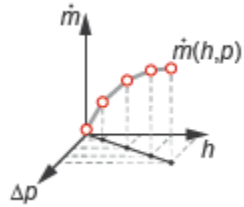
The opening area is assumed to vary linearly with the control member position. The slope of the linear expression is determined from the **Maximum valve opening** and **Maximum opening area** block parameters.



- `Tabulated data - Area vs. opening` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve. The opening area can vary nonlinearly with the control member position. The relationship between the two is given by the tabulated data in the **Valve opening vector** and **Opening area vector** block parameters.



- `Tabulated data - Mass flow rate vs. opening and pressure drop` — Calculate the mass flow rate directly from the control member position and the pressure drop across the valve. The relationship between the three variables can be nonlinear and it is given by the tabulated data in the **Valve opening vector**, **Pressure drop vector**, and **Mass flow rate table** block parameters.



Leakage Flow

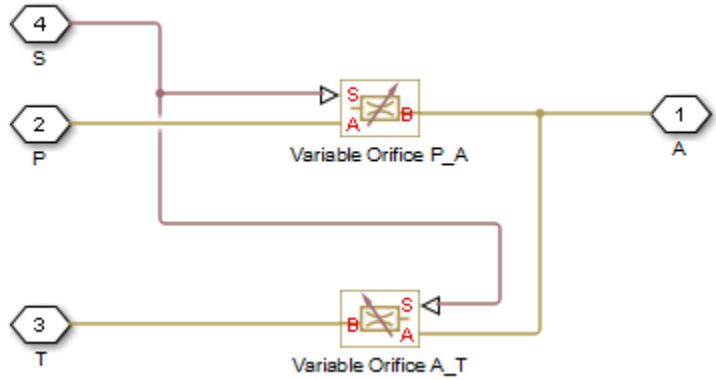
The primary purpose of the leakage flow rate of a closed valve is to ensure that at no time a portion of the thermal liquid network becomes isolated from the remainder of the model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage flow is generally present in real valves but in a model its exact value is less important than its being a small number greater than zero. The leakage flow rate is determined from the **Leakage area** block parameter.

Pressure Loss and Recovery

The pressure drop in the valve is calculated from an empirical parameter known as the discharge coefficient (obtained from the **Discharge coefficient** block parameter). The calculation captures the effect of flow regime, with the pressure drop being proportional to the mass flow rate when the flow is laminar and to the square of the same when the flow is turbulent. Also captured is the pressure recovery than in real valves occurs between the vena contracta (the narrowest aperture of the valve) and the outlet, which generally lies a small distance away.

Composite Component Structure

This block is a composite component comprising two Variable Area Orifice (TL) blocks connected as shown in the figure. A single control signal actuates the two blocks simultaneously. The **Orifice orientation** block parameters are set so that a positive signal acts to open one orifice while closing the other. The opening offsets are applied to the appropriate blocks. Refer to the Variable Area Orifice (TL) block for detail on the opening area calculations.



Ports

Input

S — Control member displacement, `unitless`

physical signal

Instantaneous displacement of the valve control member.

Conserving

P — Flow passage

thermal liquid

Opening through which the flow can enter or exit the valve. This port is typically connected to a fluid supply line.

T — Flow passage

thermal liquid

Opening through which the flow can enter or exit the valve. This port is typically connected to a fluid return line.

A — Flow passage

thermal liquid

Opening through which the flow can enter or exit the valve. This port is typically connected to an actuation line.

Parameters

Basic Parameters

`Area characteristics` — Choice of identical or different orifice opening characteristics

`Identical for all flow paths` (default) | `Different for each flow path`

Assumption to make in the opening characteristics of the valve orifices. The default setting attributes the same opening characteristics to each orifice. The maximum opening area and the orifice opening at which it occurs are then the same for all orifices. Use the alternate setting to specify these parameters separately for each orifice.

`Valve parameterization` — Method by which to model the opening characteristics of the valve

`Linear area-opening relationship` (default) | `Tabulated data - Area vs. opening` | `Tabulated data - Mass flow rate vs. opening and pressure drop`

Method by which to model the opening characteristics of the valve. The default setting prescribes a linear relationship between the valve opening area and the valve opening. The alternative settings allow for a general, nonlinear relationship to be specified in tabulated form, in one case between the opening area and the control member position (the valve opening), in the other case between the mass flow rate and both the valve opening and the pressure drop between the ports.

`Leakage area` — Opening area of the valve in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Cross-sectional area at ports A, P, and T` — Flow area at the thermal liquid ports
`0.01 m` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the valve connects.

`Characteristic longitudinal length` — Measure of the length of the flow path through the valve
`0.1` (default) | positive scalar in units of length

Average distance traversed by the fluid as it travels from inlet to outlet. This distance is used in the calculation of the internal thermal conduction that occurs between the two ports (as part of the smoothed upwind energy scheme employed in the thermal liquid domain).

`Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates
`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes
`12` (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

Model Parameterization

`Maximum valve opening` — Control member position at which the opening area is at a maximum
`5e-3 m` (default) | positive scalar in units of length

Value of the orifice opening at which an orifice is considered to be fully open. The opening area of the orifice is then at a maximum. The maximum opening is used to calculate the slope of the linear expression relating the orifice opening area to the orifice opening.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Maximum opening area` — Opening area of the valve in the fully open position

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of an orifice when its orifice opening has the value specified in the **Maximum valve opening** block parameter. The maximum opening area parameter is used to calculate the slope of the linear expression relating the orifice opening area to the orifice opening.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Smoothing factor` — Measure of the amount of smoothing to apply to the opening area function

`0.01` (default) | positive unitless scalar

Measure of the amount of smoothing to apply to the opening area function. This parameter determines the widths of the regions to be smoothed, one being at the fully open position, the other at the fully closed position. The smoothing superposes on the linear opening area function two nonlinear segments, one for each region of smoothing. The greater the value specified, the greater the smoothing and the broader the nonlinear segments.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Valve opening vector` — Vector of control member positions at which to tabulate the valve opening areas

`[-0.002, 0, 0.002, 0.005, 0.015] m` (default) | vector of positive numbers in units of length

Vector of control member positions at which to specify—dependent on the valve parameterization—the opening area of the valve or its mass flow rate. The vector elements must increase monotonically from left to right. This vector must be equal in size

to that specified in the **Opening area vector** block parameter or to the number of rows in the **Mass flow rate table** block parameter.

This data serves to construct a one-way lookup table by which to determine, from the control member position, the opening area of the valve or a two-way lookup table by which to determine, from the control member position and pressure drop, the mass flow rate of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Opening area vector` — Vector of opening areas at the specified valve opening breakpoints

`[1e-09, 2.0352e-07, 4.0736e-05, 0.00011438, 0.00034356] m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Valve opening vector** block parameter. The vector elements must increase monotonically from left to right (with increasing control member position). This vector must be equal in size to the number of valve opening breakpoints.

This data serves to construct a one-way lookup table by which to determine from the control member position the opening area of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Pressure drop vector` — Vector of pressure drops at which to tabulate the mass flow rates

`[-0.1, 0.3, 0.5, 0.7] MPa` (default) | vector in units of pressure

Vector of pressure differentials from port **A** to port **B** at which to specify the mass flow rate of the valve. The vector elements must increase monotonically from left to right. This vector must be equal in size to the number of columns in the **Mass flow rate table** block parameter.

This data serves to construct a two-way lookup table by which to determine, from the control member position and pressure drop, the opening area of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Mass flow rate table` — Matrix of mass flow rates at the specified valve opening and pressure drop breakpoints

`[-1e-05, 1.7e-05, 2e-05, 2.6e-05; -.002, .0035, .0045, .0053; -.4, .7, .9, 1.06; -1.13, 1.96, 2.5, 3; -3.45, 6, 7.7, 9.13] kg/s` (default) | matrix in units of mass/time

Matrix of mass flow rates corresponding to the breakpoints defined in the **Valve opening vector** and **Pressure drop vector** block parameters. The control member position increases from row to row from top to bottom. The pressure drop increases from column to column from left to right. The mass flow rate must increase monotonically in the same directions (with increasing control member position and increasing pressure drop).

This data serves to construct a two-way lookup table by which to determine, from the control member position and pressure drop, the opening area of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range). Ensure that the number of rows is equal to the size of the **Opening area vector** block parameter and that the number of columns is equal to the size of the **Pressure drop vector** block parameter.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow temperature` — Inlet temperature at which to specify the tabulated data

`293.15 K` (default) | positive scalar in units of temperature

Nominal inlet temperature, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the temperature measured during simulation.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow pressure` — Inlet pressure at which to specify the tabulated data

`0.101325 MPa` (default) | positive scalar in units of pressure

Nominal inlet pressure, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the pressure measured during simulation.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

Valve Opening Offsets

`Between P and A` — Control member offset from the centered position relative to the P-A orifice

`0 m` (default) | positive scalar in units of length

Distance by which to offset the valve control member from its centered (zero) position relative to the **P**-**A** orifice. This parameter determines the control member *displacement*, specified via port **S**, at which the **P**-**A** orifice is fully closed. Note that the control member *position* at which the valve is fully closed is always at zero. Set all opening offsets to positive values to model an underlapped valve or to negative values to model an overlapped valve.

`Between A and T` — Control member offset from the centered position relative to the A-T orifice

`0 m` (default) | positive scalar in units of length

Distance by which to offset the valve control member from its centered (zero) position relative to the **A**-**T** orifice. This parameter determines the control member *displacement*, specified via port **S**, at which the **A**-**T** orifice is fully closed. Note that the control member *position* at which the valve is fully closed is always at zero. Set all opening offsets to positive values to model an underlapped valve or to negative values to model an overlapped valve.

See Also

2-Way Directional Valve (TL) | 4-Way Directional Valve (TL) | Check Valve (TL) | Variable Area Orifice (TL)

Introduced in R2016a

3-Zone Pipe (2P)

Closed conduit for the transport of fluid between components
Library: Simscape / Fluids / Two-Phase Fluid / Pipes & Fittings

Description

The Pipe (2P) block models a vessel for the transport of fluid in liquid, vapor, or mixed phase. The fluid can change phase while inside, and different phases can coexist, each in a separate zone spanning a fraction of the pipe volume. Of these, there are three: one for liquid, another for vapor, and a third for the two-phase mixture that generally forms in between.

The zones do not intermix. They behave as though a thin membrane kept them apart lengthwise. They can, however, shrink and grow, one at the expense of the others, as phase change occurs. A zone can shrink to the point of disappearing—vapor wholly ceding its volume to liquid at low temperatures, for example—only to reappear when conditions reverse.

(Technically, for simulation purposes, a zone never truly disappears. A trace volume always remains inside it. At such levels, however, the zone is so small that it can be thought of as having completely vanished from the pipe.)

Use this block to efficiently capture, without resorting to chains of Pipe (2P) blocks, the dynamics of phase change along a pipe.

Pipe Connections

Ports **A** and **B** connect the pipe to neighboring components in a fluid network. Often, the components are sinks or sources of fluid—chambers, reservoirs, or, in power applications, actuators. A pump upstream of the pipe normally powers the flow, sustaining it against the pressure loss that inevitably occurs due to viscous friction along the pipe wall. That loss is calculated here using the Darcy-Weisbach equation.

Port **H**, on the other hand, connects the pipe to a thermal network. The connection sets the thermal boundary condition at the outer surface of the pipe wall. During simulation, that boundary condition determines whether—and how—the wall can exchange heat with the environment. The wall can be thermally insulated, have its outer temperature set, or be subjected to a heat flux from the outside—for example, due to convection, conduction, or radiation.

Pipe Wall

The wall is conceived as more than a thin interface between the fluid and the environment. It is furnished with thermal mass, separate from that of the fluid, and it therefore responds to thermal changes with a characteristic time lag. In other words, unlike quasi-steady components, which respond instantaneously to changing conditions, the wall has transient behavior reminiscent of real components.

In some pipes—namely those with thin walls made of light materials with high specific heat—the thermal dynamics of the wall can be fast enough to seem, from a modeling standpoint, instantaneous. In such cases, where the wall naturally behaves as a quasi-steady component, it can be easier to ignore its thermal mass altogether. This is possible by setting the **Wall thickness**, **Wall density**, or **Wall specific heat** block parameter to `0`.

The wall can also have fins. These protrusions serve to extend the surface area of the pipe, which has the effect of accelerating heat exchange. They can be internal to the pipe as well as external to it—the first enhancing heat exchange with the fluid, and the second that with the environment. The boost to the heat exchange rate depends on the number of fins and on their efficiencies, in which are encoded the effects of their geometries.

The fluid zones, being in different phases, and therefore also at different temperatures, generally exchange heat with the wall at different rates. To capture this difference, the wall, like the fluid, is divided into zones. (There is one each for liquid, vapor, and mixed phases.) Where the wall meets the environment, the zones share the boundary condition given at port **H**. At the fluid interface, their thermal conditions depend on the phases they each enclose.

Mass Balance

The fluid can enter and exit the pipe through ports **A** and **B**. Inside, the fluid volume is fixed, which is to say that the pipe wall neither dilates nor contracts. However, the fluid

being compressible and susceptible to phase change, and its density therefore liable to change, the mass in the pipe will generally vary with time. Because mass is always conserved, any change in its amount must come directly from the flow through the ports:

$$\frac{dM}{dt} = \dot{m}_{\mathrm{A}} + \dot{m}_{\mathrm{B}},$$

where M is fluid mass (its total across all pipe zones) and $\dot{m}$ is mass flow rate (its value at a port, denoted in the subscript). The time derivative on the left gives the mass accumulation in the pipe, computed during simulation as:

$$\frac{dM}{dt} = \left[\left(\frac{d\rho}{dp} \right)_u \frac{dp}{dt} + \left(\frac{d\rho}{du} \right)_p \frac{du}{dt} + \rho_{\mathrm{L}} \frac{dz_{\mathrm{L}}}{dt} + \rho_{\mathrm{M}} \frac{dz_{\mathrm{M}}}{dt} + \rho_{\mathrm{V}} \frac{dz_{\mathrm{V}}}{dt} \right] V,$$

where:

- ρ is density (conceived, in the last three terms, as the average over the pipe zone denoted in the subscript—`L` for liquid, `M` for mixture, and `V` for vapor).
- p is pressure.
- u is specific internal energy.
- z is the pipe length fraction occupied by a phase (denoted in the subscript).
- V is the total pipe volume, the product of the pipe length and cross-sectional area (specified in the block parameters of the same names).

The leftmost partial derivative describes how density changes with pressure; the rightmost partial derivative, how it changes with specific internal energy. Together, they capture the mass accumulation due to fluid compressibility. The last three terms give the density variation due to phase change (as reflected in the growth rates of the liquid, vapor, and mixed fluid zones).

The partial derivatives of the first two terms are calculated as sums over the different pipe zones. The data for each zone is specified in the Two-Phase Fluid Properties (2P) block (required in every two-phase fluid model). For the leftmost partial derivative, the sum is:

$$\left(\frac{\partial \rho}{\partial p} \right)_u = z_{\mathrm{L}} \left(\frac{\partial \rho}{\partial p} \right)_{u,\mathrm{L}} + z_{\mathrm{M}} \left(\frac{\partial \rho}{\partial p} \right)_{u,\mathrm{M}} + z_{\mathrm{V}} \left(\frac{\partial \rho}{\partial p} \right)_{u,\mathrm{V}}.$$

Likewise for the rightmost partial derivative:

$$\left(\frac{\partial \rho}{\partial u}\right)_p = z_{\mathrm{L}}\left(\frac{\partial \rho}{\partial u}\right)_{p,\mathrm{L}} + z_{\mathrm{M}}\left(\frac{\partial \rho}{\partial u}\right)_{p,\mathrm{M}} + z_{\mathrm{V}}\left(\frac{\partial \rho}{\partial u}\right)_{p,\mathrm{V}}.$$

The liquid, vapor, and mixed-phase partial derivatives are the averages for the respective pipe zones.

Energy Balance

The energy in the pipe can vary by flow through the ports—either by advection or conduction—and by convective heat exchange with the wall:

$$\frac{dE}{dt} = \phi_{\mathrm{A}} + \phi_{\mathrm{B}} + Q_{\mathrm{F}},$$

where:

- E is energy.
- ϕ is energy flow rate, defined here as that *into* the pipe through the ports (denoted in the subscripts).
- Q is the rate of convective heat exchange between the fluid and the wall (subscript F). That rate depends on, but is not the same as, the thermal boundary condition specified at port H. Its calculation depends on whether the thermal mass of the wall is zero or finite).

The time derivative on the left gives the rate of energy accumulation in the pipe:

$$\frac{dE}{dt} = M\frac{du_{\mathrm{O}}}{dt} + \frac{dM}{dt}u_{\mathrm{O}},$$

where u is the specific internal energy at the outlet (port **A** or port **B**, dependent on flow direction), after all heat exchange has taken place. The second time derivative is as defined in ``Mass Balance.''

Heat Exchange

The heat exchange rate between the fluid and the wall depends on the thermal mass of the wall, defined as:

$$C_{\mathrm{W}} = c_{\mathrm{p,W}} \rho_{\mathrm{W}} S_{\mathrm{W}} \left(t_{\mathrm{W}} + \frac{t_{\mathrm{W}}^2}{D_{\mathrm{H}}} \right),$$

where the subscript `W` denotes a wall parameter and:

- C is thermal mass.
- c_{p} is specific heat.
- t is thickness.
- D_{H} is the hydraulic diameter (that which a pipe of any shape would have if its cross section were made circular.
- S is surface area, here that of a wall without fins, calculated as:

  $$S_{\mathrm{W}} = \frac{4A}{D_{\mathrm{H}}} L,$$

  where A is the cross-sectional area of the pipe and L is its length.

The specific heat, density, and thickness of the wall are specified as block parameters in the **Heat Transfer** tab. The hydraulic diameter, cross-sectional area, and length of the pipe are given (also as block parameters) in the **Geometry** tab.

If the thermal mass of the wall is zero, the heat exchange between the fluid and the wall is equal to that between the fluid and the environment. (The thermal mass is zero if either the specific heat, density, or thickness of the wall is set to zero.) The rate of that exchange is:

$$Q_{\mathrm{F}} = \dot{m}_{\mathrm{H}} \left(\Delta h_{\mathrm{L}} + \Delta h_{\mathrm{M}} + \Delta h_{\mathrm{V}} \right),$$

where:

- $\dot{m}_{\mathrm{H}}$ is the mass flow rate for heat exchange calculations (an average of the values at the ports).

- Δh is the change in specific enthalpy due to heat exchange between the environment (through the wall) and the fluid zones—liquid (subscript `L`), vapor (`V`), or mixed-phase (`M`).

The change in specific enthalpy across a zone is calculated as the difference between the entrance and exit values for that zone:

$$\Delta h = h_{\mathrm{O}} - h_{\mathrm{I}},$$

where h is specific enthalpy and the subscripts `I` and `O` denote the values at the entrance (or *inlet*) of the zone and at its exit (or *outlet*). The specific enthalpy is obtained for each zone from the tabulated data of the Two-Phase Fluid Properties (2P) block.

As the zones are separated by phase change, the specific enthalpy within each zone is bounded by the saturation values of the phases present. The bounds depend on the zone. In the liquid zone, a maximum is imposed at the saturation value of the liquid phase, giving for its change in specific enthalpy:

$$\Delta h_{\mathrm{L}} = \min\left(h_{\mathrm{O}}, h_{\mathrm{S,L}}\right) - \min\left(h_{\mathrm{I}}, h_{\mathrm{S,L}}\right),$$

where the subscript `S,L` denotes the liquid saturation value. (The `min` function ensures that the specific enthalpies used in the calculation never exceed the liquid saturation value.) In the vapor zone, a minimum is imposed at the saturation value of the vapor phase, giving for its change in specific enthalpy:

$$\Delta h_{\mathrm{V}} = \max\left(h_{\mathrm{O}}, h_{\mathrm{S,V}}\right) - \max\left(h_{\mathrm{I}}, h_{\mathrm{S,V}}\right),$$

where the subscript `S,V` denotes the vapor saturation value. (The liquid and vapor saturation values will generally differ by an amount equal to the latent heat of the fluid.) In the mixed-phase zone, a maximum is imposed at the saturation value of the vapor phase and a minimum at the saturation value of the liquid phase, giving the (slightly more complicated) expression:

$$\Delta h_{\mathrm{V}} = \min\left[\max\left(h_{\mathrm{O}}, h_{\mathrm{S,L}}\right), h_{\mathrm{S,V}}\right] - \min\left[\max\left(h_{\mathrm{I}}, h_{\mathrm{S,L}}\right), h_{\mathrm{S,V}}\right].$$

The change in specific enthalpy across a zone gives the length fraction of the same. In the liquid zone, the relationship between the two variables—Δh_L and z_L—is given by the expression:

$$\Delta h_L = c_{p,L}\left(T_E - T_I\right)\left[1 - \exp\left(-\frac{z_L S_W}{\dot{m}_Q c_{p,L}\left(\alpha_L^{-1} + \alpha_E^{-1}\right)}\right)\right],$$

where the subscript L denotes an average value over the liquid zone, subscript E one in the environment (as measured at the outer wall surface), and I one at the entrance of the liquid zone, before any heat exchange has had a chance to occur. The mass flow rate $\dot{m}_Q$ is an average taken from the ports for use in heat exchange calculations. For the remaining equation variables:

- T is temperature. (That at the entrance of the liquid zone, T_I, is limited to a maximum given by the saturation point, at which liquid will generally coexist with vapor, in the mixed-phase zone of the pipe.)
- α is the heat transfer coefficient—at the wall-fluid interface in the liquid zone (subscript L) or at the wall-environment interface (subscript E). Their values are calculated as described in "Zone Heat Transfer Coefficients."

Likewise for the vapor zone:

$$\Delta h_V = c_{p,V}\left(T_E - T_I\right)\left[1 - \exp\left(-\frac{z_V S_W}{\dot{m}_Q c_{p,V}\left(\alpha_V^{-1} + \alpha_E^{-1}\right)}\right)\right],$$

where the subscript V denotes an average value over the vapor zone. The temperature at the entrance of the vapor zone, T_V, is limited to a minimum given by the saturation point, at which vapor will generally coexist with liquid (in the mixed-phase zone).

Lastly, for the mixed-phase zone:

$$\Delta h_M = \left(T_E - T_S\right)\frac{z_M S_W}{\dot{m}_Q\left(\alpha_M^{-1} + \alpha_E^{-1}\right)},$$

where the subscript `M` denotes an average value over the mixed-phase zone and the subscript `S` denotes a saturation value. Implicit in this expression is the fact that the mixed-phase zone is always at the fluid saturation temperature.

The heat transfer coefficient of a zone follows from the definition of the Nusselt number—the ratio of convective to conductive heat exchange rates. In a (general) zone without wall fins:

$$\mathrm{Nu} = \frac{\alpha D_{\mathrm{H}}}{k},$$

where `Nu` is the Nusselt number of the zone, k its average thermal conductivity, and D_{H} (the hydraulic diameter of the pipe) the characteristic length for convective heat transfer in the zone. The thermal conductivity depends on the phase enclosed in the zone—liquid, vapor, or a mixture. Its value is obtained during simulation from the tabulated functions of the Two-Phase Fluid Properties (2P) block.

Adding fins to the inner surface of the wall enhances its convective heat exchange with the fluid such that:

$$\mathrm{Nu}\left(1+\eta_{\mathrm{Int}} s_{\mathrm{Int}}\right) = \frac{\alpha D_{\mathrm{H}}}{k},$$

where η_{Int} is the efficiency of the internal fins—a fraction, normally between `0` and `1`, in which are encoded the effects of fin geometry—and s_{Int} is the ratio of the inner wall surface area with fins to that without. The internal fin efficiency and the surface area ratio are specified as block parameters in the **Heat Transfer** tab.

In the laminar regime, the Nusselt number is a constant, specified in the **Nusselt number for laminar flow heat transfer** block parameter. The default parameter value, `3.66`, corresponds to a constant temperature at the inner wall surface. When the flow grows turbulent, the Nusselt number begins to increase, with its value being computed from empirical correlations—the choice of which depends on the zone.

Rearranging terms and capturing the dependence on flow regime in the calculation, gives for the (general form of the) heat transfer coefficient:

$$\alpha_{*} = \max\left(\mathrm{Nu}_{*,\mathrm{Tur}}, \mathrm{Nu}_{\mathrm{Lam}}\right)\left(1+\eta_{\mathrm{Int}} s_{\mathrm{Int}}\right)\frac{k_{*}}{D_{\mathrm{H}}},$$

where the asterisk in the subscript denotes a value specific to the zone considered.

The heat transfer coefficient for thermal exchanges between the outer surface of the wall and its external surroundings, denoted here α_{Ext}, is specified explicitly in the block (via the **External environment heat transfer coefficient** block parameter). If the thermal mass of the wall is ignored and the wall has no fins, this parameter equals the heat transfer coefficient of the environment, α_{Ext}, used in the "Zone Length Fraction" calculations.

Adding fins to the outer surface of the wall enhances its convective heat exchange with the environment such that:

$$\alpha_E = \alpha_{Ext}\left(1+\eta_{Ext}s_{Ext}\right),$$

where the subscript denotes a value specific to the external fins—η being their efficiency and s the ratio of surface area with external fins to that without.

If the thermal mass of the wall is finite, then thermal conduction through the wall will act to lower the heat transfer coefficient between the inner wall surface and the environment:

$$\frac{1}{\alpha_E} = \frac{1}{\alpha_W} + \frac{1}{\alpha_{Ext}\left(1+\eta_{Ext}s_{Ext}\right)},$$

where α_W is the heat transfer coefficient due to thermal conduction between the inner and outer surfaces of the wall:

$$\alpha_W = \frac{k_W}{D_H \ln\left(1+\frac{t_W}{D_H}\right)}$$

In the liquid and vapor zones, the Nusselt number for turbulent heat exchange follows from the Gnielinski correlation, which, in its general form, gives:

$$\mathrm{Nu}_{*,\mathrm{Tur}} = \frac{f_*/8\left(\mathrm{Re}_* - 1000\right)\mathrm{Pr}_*}{1+12.7\left(f_*/8\right)^{1/2}\left(\mathrm{Pr}_*^{2/3}-1\right)},$$

where Re is the average Reynolds number in the fluid zone, Pr is the average Prandtl number in the same, and f is the Darcy friction factor, calculated from the Haaland correlation:

$$\frac{1}{\sqrt{f_*}} = -1.8\log\left[\left(\frac{/D_\mathrm{H}}{3.7}\right)^{1.11} + \frac{6.9}{\mathrm{Re}_*}\right],$$

where ε is the absolute roughness of the pipe, a measure of the height of the bumps at the pipe-fluid interface; typical roughness values range from 0.0015 mm for certain plastic and glass tubes to 3 mm for larger concrete pipes. The average Reynolds number is:

$$\mathrm{Re}_* = \frac{\dot{m}_\mathrm{Q} D_\mathrm{H}}{\mu_* S},$$

where μ^* is the average dynamic viscosity of the fluid zone.

In the mixed-phase zone, the Nusselt number for turbulent heat exchange follows from the Cavallini and Zecchin correlation. Averaged over the change in vapor quality across the pipe, the correlation gives:

$$\mathrm{Nu}_\mathrm{M,Tur} = \frac{1}{x_\mathrm{O} - x_\mathrm{I}} \int_{x_\mathrm{I}}^{x_\mathrm{O}} 0.05\left(1 - x + x\sqrt{\frac{\nu_\mathrm{L,Sat}}{\nu_\mathrm{V,Sat}}}\right)^{0.8} \mathrm{Re}_\mathrm{L,Sat}^{0.8}\,\mathrm{Pr}_\mathrm{L,Sat}^{0.33}\,\mathrm{d}x,$$

where the Reynolds and Prandtl numbers are the average values in the saturated liquid—calculated, in the case of the Reynolds number, from the dynamic viscosity of the saturated liquid—and:

- x is vapor quality—at the inlet, before heat exchange has taken place (subscript I), or at the outlet, after all heat transfer has occurred (subscript O).
- ν is specific volume—of the saturated liquid (subscript L,Sat) or of the saturated vapor (subscript V,Sat).

The final, integrated form of the Cavallini and Zecchin correlation is:

$$\mathrm{Nu}_{\mathrm{M,Tur}} = \frac{0.05\mathrm{Re}_{\mathrm{L,Sat}}^{0.8}\mathrm{Pr}_{\mathrm{L,Sat}}^{0.33}\left(\left[\left(\sqrt{\frac{\nu_{\mathrm{V,Sat}}}{\nu_{\mathrm{L,Sat}}}-1}\right)x_{\mathrm{O}}+1\right]^{1.8}-\left[\left(\sqrt{\frac{\nu_{\mathrm{V,Sat}}}{\nu_{\mathrm{L,Sat}}}-1}\right)x_{\mathrm{I}}+1\right]^{1.8}\right)}{1.8\left(\sqrt{\frac{\nu_{\mathrm{V,Sat}}}{\nu_{\mathrm{L,Sat}}}-1}\right)\left(x_{\mathrm{O}}-x_{\mathrm{I}}\right)}$$

If the thermal mass of the wall is finite, the heat exchange between the fluid and the environment must reflect also the thermal conduction through the wall. The effect of wall conduction will vary between zones and must be considered separately for each. The total heat exchange rate between the fluid and the inner surface of the wall, Q_{F}, featured in the energy balance of the fluid (see "Energy Balance"), is the sum of the contributions from the separate zones—$Q_{\mathrm{F,L}}$, $Q_{\mathrm{F,V}}$, and $Q_{\mathrm{F,M}}$:

$$Q_{\mathrm{F}} = Q_{\mathrm{F,L}} + Q_{\mathrm{F,V}} + Q_{\mathrm{F,M}},$$

The individual heat exchange rates $Q_{\mathrm{F},*}$ are each constrained by the energy balance for the wall section enclosing the fluid zone. For the liquid zone, that balance takes the form:

$$C_{\mathrm{W}}\left[z_{\mathrm{L}}\frac{dT_{\mathrm{W,L}}}{dt}+\max\left(\frac{dz_{\mathrm{L}}}{dt},0\right)\left(T_{\mathrm{W,L}}-T_{\mathrm{W,M}}\right)\right]=Q_{\mathrm{H,L}}-Q_{\mathrm{F,L}},$$

where Q_{H} is the heat exchange rate between the environment and the fluid (here, that in the liquid zone). Likewise for the vapor zone:

$$C_{\mathrm{W}}\left[z_{\mathrm{V}}\frac{dT_{\mathrm{W,V}}}{dt}+\max\left(\frac{dz_{\mathrm{V}}}{dt},0\right)\left(T_{\mathrm{W,V}}-T_{\mathrm{W,M}}\right)\right]=Q_{\mathrm{H,V}}-Q_{\mathrm{F,V}}.$$

Lastly, for the mixed zone:

$$C_{\mathrm{W}}\left[z_{\mathrm{M}}\frac{dT_{\mathrm{W,M}}}{dt}+\min\left(\frac{dz_{\mathrm{L}}}{dt},0\right)\left(T_{\mathrm{W,L}}-T_{\mathrm{W,M}}\right)+\min\left(\frac{dz_{\mathrm{V}}}{dt},0\right)\left(T_{\mathrm{W,V}}-T_{\mathrm{W,M}}\right)\right]=Q_{\mathrm{H,M}}-Q_{\mathrm{F,M}},$$

The heat exchange rate between the fluid and the inner surface of the pipe wall in the liquid zone is:

$$Q_{\mathrm{F,L}} = \dot{m}_{\mathrm{Q}} c_{\mathrm{p,L}} \left[T_{\mathrm{W,L}} - \min\left(T_{\mathrm{I}}, T_{\mathrm{Sat}}\right) \right] \left[1 - \exp\left(- \frac{z_{\mathrm{L}} S_{\mathrm{W}} \alpha_{\mathrm{L}}}{\dot{m}_{\mathrm{Q}} c_{\mathrm{p,L}}} \right) \right].$$

Likewise for the vapor zone:

$$Q_{\mathrm{F,V}} = \dot{m}_{\mathrm{Q}} c_{\mathrm{p,V}} \left[T_{\mathrm{W,V}} - \min\left(T_{\mathrm{I}}, T_{\mathrm{Sat}}\right) \right] \left[1 - \exp\left(- \frac{z_{\mathrm{V}} S_{\mathrm{W}} \alpha_{\mathrm{V}}}{\dot{m}_{\mathrm{Q}} c_{\mathrm{p,V}}} \right) \right].$$

Lastly for the mixed-phase zone:

$$Q_{\mathrm{F,M}} = \left(T_{\mathrm{H}} - T_{\mathrm{Sat}}\right) z_{\mathrm{M}} S_{\mathrm{W}} \alpha_{\mathrm{M}}$$

The overall heat exchange between the fluid in the liquid zone and the environment is defined as:

$$Q_{\mathrm{H,L}} = \left(T_{\mathrm{H}} - T_{\mathrm{W,L}}\right) z_{\mathrm{L}} S_{\mathrm{W}} \alpha_{\mathrm{E}}.$$

Likewise for the vapor zone:

$$Q_{\mathrm{H,L}} = \left(T_{\mathrm{H}} - T_{\mathrm{W,L}}\right) z_{\mathrm{L}} S_{\mathrm{W}} \alpha_{\mathrm{E}}.$$

Lastly for the mixed-phase zone:

$$Q_{\mathrm{H,M}} = \left(T_{\mathrm{H}} - T_{\mathrm{W,M}}\right) z_{\mathrm{M}} S_{\mathrm{W}} \alpha_{\mathrm{E}}.$$

Momentum Balance

For its pressure calculations, the pipe is conceived as an assembly of two control volumes. One lies between port **A** and the interior of the pipe; the other, between the interior of the

pipe and port **B**. In both, the inertial force on the fluid is ignored—a suitable approximation at the large time scales over which changes to flow typically occur. The pipe is assumed to lie flat and the elevation head between its ports is therefore zero in both volumes.

In the half pipe next to port **A**, the momentum balance is:

$$p_{\mathrm{A}} - p_{\mathrm{I}} = \left(\frac{1}{\rho_{\mathrm{I}}} - \frac{1}{\rho_{\mathrm{A}}^{*}}\right)\left(\frac{\dot{m}_{\mathrm{A}}}{S}\right)^{2} + p_{\mathrm{F,A}},$$

where $p_{\mathrm{F,A}}$ is the pressure loss due to friction in the half pipe and the subscript **I** denotes an average value in the interior of the pipe. The asterisk denotes a value for the momentum flux term (first on the right). The average fluid density in the pipe is a function of the densities and length fractions of the fluid zones:

$$\rho = \rho_{\mathrm{L}} z_{\mathrm{L}} + \rho_{\mathrm{M}} z_{\mathrm{M}} + \rho_{\mathrm{V}} z_{\mathrm{V}}.$$

Heat transfer is assumed to occur entirely at the pressure inside the pipe (p_{I}). For pressure calculations, the pipe half is assumed to be adiabatic, and the momentum flux term is therefore constrained by the adiabatic expression:

$$u_{\mathrm{A}}^{*} + \frac{p_{\mathrm{A}}}{\rho_{\mathrm{A}}^{*}} + \frac{1}{2}\left(\frac{\dot{m}_{\mathrm{A}}}{\rho_{\mathrm{A}}^{*} S}\right)^{2} = h + \frac{1}{2}\left(\frac{\dot{m}_{\mathrm{A}}}{\rho S}\right)^{2},$$

where the specific enthalpy (h) is the average over the zones in the pipe:

$$h = h_{\mathrm{L}} z_{\mathrm{L}} + h_{\mathrm{V}} z_{\mathrm{V}} + h_{\mathrm{M}} z_{\mathrm{M}}.$$

Likewise for the half pipe next to port **B**, the momentum balance is:

$$p_{\mathrm{B}} - p_{\mathrm{I}} = \left(\frac{1}{\rho_{\mathrm{I}}} - \frac{1}{\rho_{\mathrm{B}}^{*}}\right)\left(\frac{\dot{m}_{\mathrm{B}}}{S}\right)^{2} + p_{\mathrm{F,B}}.$$

The momentum flux term is constrained by the adiabatic expression:

$$u_{\mathrm{B}}^{*}+\frac{p_{\mathrm{B}}}{\rho_{\mathrm{B}}^{*}}+\frac{1}{2}\left(\frac{\dot{m}_{\mathrm{B}}}{\rho_{\mathrm{B}}^{*}S}\right)^{2}=h+\frac{1}{2}\left(\frac{\dot{m}_{\mathrm{B}}}{\rho S}\right)^{2}$$

The calculation of the major pressure loss (due to friction in the pipe) is based on the Darcy-Weisbach equation:

$$p_{\mathrm{F},j}=f_{\mathrm{D},j}\,\frac{1}{4}\,\frac{L_{\mathrm{E}}\dot{m}_{j}\left|\dot{m}_{j}\right|}{\rho_{\mathrm{I}}D_{\mathrm{H}}S^{2}},$$

where f_{D} is the Darcy friction factor and the subscript j denotes the pipe half—that adjacent to port **A** or to port **B**. L_{E} is the effective length of the pipe and D_{H} is the hydraulic diameter of the same. The effective pipe length is the sum of the true pipe length (L) and the aggregate equivalent length of all local resistances (L_{R}), such as elbows, fittings, and unions:

$$L_{\mathrm{E}}=L+L_{\mathrm{R}},$$

When the flow is laminar, the friction factor (for a given pipe geometry) is a function of the Reynolds number alone:

$$f_{\mathrm{D},j}=\frac{\lambda}{\mathrm{Re}_{j}},$$

where λ is the shape factor of the pipe, an empirical constant used to encode the effect of pipe geometry on the major pressure loss. Its value is specified in the **Shape factor for laminar viscous friction** block parameter. Typical values include 64 in circular pipes and 48–96 in noncircular ones. The Reynolds number at port k is defined as:

$$\mathrm{Re}_{j}=\frac{\dot{m}_{j}D_{\mathrm{H}}}{\mu_{\mathrm{I}}S},$$

where μ is the average dynamic viscosity in the pipe, a function of the dynamic viscosities and length fractions of the fluid zones:

$$\mu=\mu_{\mathrm{L}}z_{\mathrm{L}}+\mu_{\mathrm{V}}z_{\mathrm{M}}+\mu_{\mathrm{V}}z_{\mathrm{V}},$$

The dynamic viscosities are obtained for each zone from the Two-Phase Fluid Properties (2P) block. The actual pressure loss calculation in the laminar flow regime is carried out for each pipe half as:

$$p_{\mathrm{F},j} = \frac{1}{4}\frac{\lambda \mu_{\mathrm{I}} L_{\mathrm{E}} \dot{m}_j}{\rho_{\mathrm{I}} D_{\mathrm{H}}^2 S},$$

When the flow is turbulent, the friction factor is instead calculated from the Haaland correlation, which, expressed for each pipe half, gives:

$$\frac{1}{\sqrt{f_{\mathrm{D},j}}} = -1.8\log\left[\left(\frac{\varepsilon/D}{3.7}\right)^{1.11} + \frac{6.9}{\mathrm{Re}_j}\right].$$

In transitional flows—those at Reynolds numbers too large to be laminar and too small to be turbulent—the pressure loss due to friction is smoothed between the two regimes. The smoothing is based on a nonlinear expression and it extends over the range between the **Laminar flow upper Reynolds number limit** and **Turbulent flow lower Reynolds number limit** block parameters.

Ports

Output

Z — Zone length fractions

physical signal

Vector with the length fractions of the liquid, vapor, and mixed-phase zones in the pipe.

Conserving

A — Pipe opening

two-phase fluid

Opening through which the thermal liquid flows into or out of the pipe. Ports **A** and **B** can each function as either inlet or outlet. Thermal conduction is allowed between the

thermal liquid ports and the fluid internal to the pipe (though its impact is typically relevant only at near zero flow rates).

`B` — Pipe opening
two-phase fluid

Opening through which the thermal liquid flows into or out of the pipe. Ports **A** and **B** can each function as either inlet or outlet. Thermal conduction is allowed between the thermal liquid ports and the fluid internal to the pipe (though its impact is typically relevant only at near zero flow rates).

`H` — Pipe wall
thermal

Thermal boundary condition at the outer surface of the pipe wall. Use this port to capture heat exchange of various kinds—for example, conductive, convective, or radiative—between the pipe wall and the environment. Heat exchange between the inner surface of the wall and the fluid is captured directly in the block.

Parameters

Geometry

`Pipe length` — Distance between the ports of the pipe
`5 m` (default) | positive scalar with units of length

Distance between the ports of the pipe. The liquid, vapor, and two-phase zones each comprise a fraction of this distance. The zone fractions can vary but their aggregate length, being the same as the distance of the pipe, is fixed.

`Cross-sectional area` — Internal area normal to the direction of the flow
`0.01 m^2` (default) | positive scalar with units of area

Internal area of the pipe normal to the direction of flow. The cross section of the pipe is assumed to be constant throughout its length.

`Hydraulic diameter` — Diameter of a cylindrical pipe with the given cross-sectional area
`0.1 m` (default) | positive scalar in units of length

Ratio of the opening area of a cross section of the pipe to the perimeter of that area. The pipe is not required to be cylindrical, and its cross-section have any shape. This parameter gives the diameter that a general cross section would have if it were circular.

Viscous Friction

`Aggregate equivalent length of local resistances` — Minor pressure loss in the pipe expressed as a length
`1 m` (default) | positive scalar in units of length

Combined length of all local resistances present in the pipe. Local resistances include bends, fittings, armatures, and pipe inlets and outlets. The effect of the local resistances is to increase the effective length of the pipe. This length is added to the geometrical pipe length for friction calculations.

`Internal surface absolute roughness` — Average depth of surface defects on the internal surface of the pipe
`15e-6 m` (default) | positive scalar in units of length

Average depth of all surface defects on the internal surface of the pipe. The surface defects affect the pressure loss across the pipe in the turbulent flow regime.

`Laminar flow upper Reynolds number limit` — Reynolds number below which the flow is laminar
`2e+3` (default) | positive unitless scalar

Reynolds number above which the pipe flow begins to transition from laminar to turbulent. This value is the maximum Reynolds number corresponding to fully developed laminar flow.

`Turbulent flow lower Reynolds number limit` — Reynolds number above which the flow is turbulent
`4e+3` (default) | positive unitless scalar

Reynolds number below which the pipe flow begins to transition from turbulent to laminar. This value is the minimum Reynolds number corresponding to fully developed turbulent flow.

`Shape factor for laminar flow viscous friction` — Dimensionless factor used to capture the effect of geometry on viscous friction losses
`64` (default) | positive unitless scalar

Dimensionless factor used to capture the effects of cross-sectional geometry on the viscous friction losses incurred in the laminar flow regime. Typical values are `64` for a circular cross section, `57` for a square cross section, and `62` for a rectangular cross section with an aspect ratio of 2.

Heat Transfer

`External environment heat transfer coefficient` — Heat transfer coefficient between the environment and the pipe wall
`100 W/(m^2*K)` (default) | positive scalar with units of power/(area*temperature)

Heat transfer coefficient for heat exchange between the environment (at port **H**) and the outer surface of the pipe wall. If this parameter is set to `inf`, then the thermal resistance between the environment and the wall is assumed to be zero. The pipe wall then has a uniform temperature equal to that associated with port **H**.

`Wall thickness` — Average thickness of the pipe wall material
`0 m` (default) | positive scalar with units of length

Average thickness of the pipe wall material. If this value is set to `0`, then both thermal resistance due to conduction through the pipe wall and thermal storage due to the thermal mass of the pipe wall are assumed to be negligible.

`Wall thermal conductivity` — Thermal conductivity of the pipe wall material
`390 W/(m*K)` (default) | positive scalar with units of power/(length*temperature)

Thermal conductivity of the pipe wall material. If this parameter is set to `inf`, then thermal resistance due to conduction through the pipe wall is assumed to be negligible.

`Wall specific heat` — Heat capacity per unit mass of pipe wall material
`390 J/kg/K` (default) | positive scalar with units of energy/(mass*temperature)

Heat capacity per unit mass of the pipe wall material. If this parameter is set to `0`, then thermal storage due to the thermal mass of the pipe is assumed to be negligible.

`Wall density` — Mass density of the pipe wall material
`9e3 kg/m^3` (default) | positive scalar with units of mass/volume

Mass density of the pipe wall material. If this parameter is set to `0`, then thermal storage due to the thermal mass of the pipe wall is assumed to be negligible.

`Ratio of external fins surface area to no-fin surface area` — Ratio of heat transfer surface area with external fins to that without
`0` (default) | positive unitless scalar

Ratio of the total heat transfer surface area of the fins on the external side of the pipe wall to that of the pipe wall without any fins. The presence of fins serves to enhance the convective heat transfer between the pipe wall and the environment.

`External fin efficiency` — Ratio of actual to ideal heat exchange rates with external fins
`0.5` (default) | positive unitless scalar

Ratio of the actual heat exchange rate between the external fins and the environment to its ideal value (if the fins were entirely held at the temperature of the pipe wall). This parameter is a function of fin geometry.

`Ratio of internal fins surface area to no-fin surface area` — Ratio of heat transfer surface area with internal fins to that without
`0` (default) | positive unitless scalar

Ratio of the total heat transfer surface area of the fins on the internal side of the pipe wall to that of the pipe wall without any fins. The presence of fins serves to enhance the convective heat transfer between the pipe wall and the fluid.

`Internal fin efficiency` — Ratio of actual to ideal heat exchange rates with internal fins
`0.5` (default) | positive unitless scalar

Ratio of the actual heat exchange rate between the internal fins and the environment to its ideal value (if the fins were entirely held at the temperature of the pipe wall). This parameter is a function of fin geometry.

`Nusselt number for laminar flow heat transfer` — Ratio of convective to conductive heat exchange rates in the laminar flow regime
`3.66` (default)

Ratio of convective to conductive heat exchange rates in the laminar flow regime. This parameter is a function of pipe cross section geometry. Typical values are `3.66` for a circular cross section, `2.98` for a square cross section, and `3.39` for a rectangular cross section with an aspect ratio of 2.

Effects and Initial Conditions Tab

`Initial fluid energy specification` — Thermodynamic variable whose initial value to set
`Temperature` (default) | `Vapor quality` | `Vapor void fraction` | `Specific enthalpy` | `Specific internal energy`

Thermodynamic variable in terms of which to define the initial conditions of the component.

`Initial pressure` — Absolute pressure at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure in the pipe at the start of simulation, specified against absolute zero.

`Initial temperature` — Absolute temperature at the start of simulation
`293.15 K` (default) | scalar or two-element vector with units of temperature

Fluid temperature in the pipe at the start of simulation. This parameter can be a scalar or a 2-element vector. If it is a scalar, the initial temperature is assumed to be uniform throughout the pipe. If it is a vector, the initial temperature is assumed to vary linearly between the ports. The first vector element gives the initial temperature at the inlet and the second vector element that at the outlet.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Temperature`.

`Initial vapor quality` — Mass fraction of vapor at the start of simulation
`0.5` (default) | unitless scalar between 0 and 1

Vapor quality, or mass fraction of vapor, in the pipe at the start of simulation. This parameter can be a scalar or a 2-element vector. If it is a scalar, the initial vapor quality is assumed to be uniform throughout the pipe. If it is a vector, the initial vapor quality is assumed to vary linearly between the ports. The first vector element gives the initial vapor quality at the inlet and the second vector element that at the outlet.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Vapor quality`.

`Initial vapor void fraction` — Volume fraction of vapor at the start of simulation

`0.5` (default) | unitless scalar between 0 and 1

Vapor void fraction, or vapor volume fraction, in the pipe at the start of simulation. This parameter can be a scalar or a 2-element vector. If it is a scalar, the initial vapor void fraction is assumed to be uniform throughout the pipe. If it is a vector, the initial vapor void fraction is assumed to vary linearly between the ports. The first vector element gives the initial vapor void fraction at the inlet and the second vector element that at the outlet..

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Vapor void fraction`.

`Initial specific enthalpy` — Specific enthalpy at the start of simulation

`1500 kJ/kg` (default) | scalar with units of energy/mass

Specific enthalpy of the fluid in the pipe at the start of simulation. This parameter can be a scalar or a 2-element vector. If it is a scalar, the initial specific enthalpy is assumed to be uniform throughout the pipe. If it is a vector, the initial specific enthalpy is assumed to vary linearly between the ports. The first vector element gives the initial specific enthalpy at the inlet and the second vector element that at the outlet.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Specific enthalpy`.

`Initial specific internal energy` — Specific internal energy at the start of simulation

`1500 kJ/kg` (default) | scalar with units of energy/mass

Specific internal energy of the fluid in the pipe at the start of simulation. This parameter can be a scalar or a 2-element vector. If it is a scalar, the initial specific internal energy is assumed to be uniform throughout the pipe. If it is a vector, the initial specific internal energy is assumed to vary linearly between the ports. The first vector element gives the initial specific internal energy at the inlet and the second vector element that at the outlet.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Specific internal energy`.

`Time constant for change in zone length fraction` — Characteristic time scale for changes in fluid zone lengths
`0.1 s` (default) | scalar with units of time

Characteristic time scale for changes in the lengths of the fluid zones. Increase this parameter to slow the rate of phase change or decrease it to speed up the rate.

References

[1] White, F.M., *Fluid Mechanics*, 7th Ed, Section 6.8. McGraw-Hill, 2011.

[2] Cengel, Y.A., *Heat and Mass Transfer—A Practical Approach*, 3rd Ed, Section 8.5. McGraw-Hill, 2007.

See Also

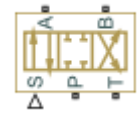
Receiver Accumulator (2P)

Introduced in R2018b

4-Way Directional Valve

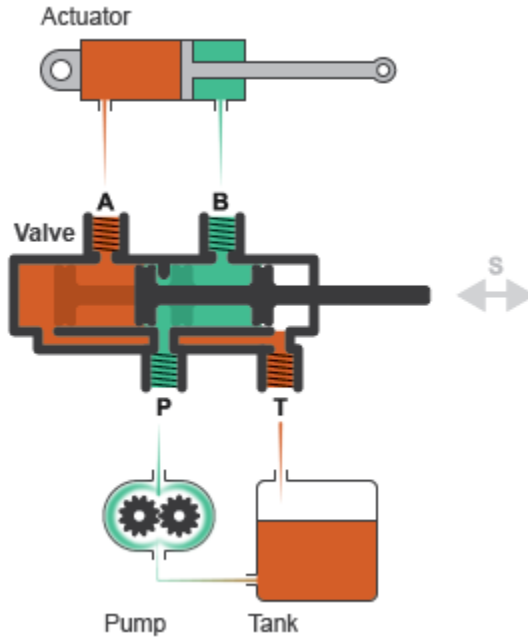
Four-port three-position directional control valve

Library: Simscape / Fluids / Hydraulics (Isothermal) / Valves / Directional Valves



Description

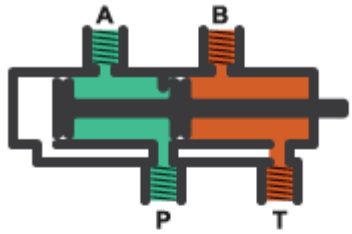
The 4-Way Directional Valve block represents a directional control valve with four ports and three positions, or flow paths. The ports connect to what in a typical model are a hydraulic pump (port **P**), a storage tank (port **T**), and a double-acting actuator (ports **A** and **B**). Fluid can flow from the pump to the actuator via path **P-A** or **P-B** and from the actuator to the tank via path **A-T** or **B-T**—depending on the working side of the actuator.



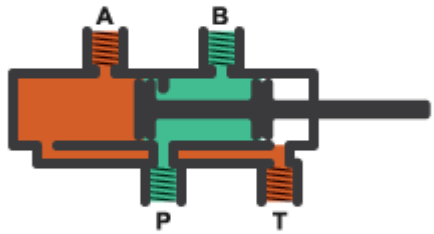
Typical Valve Setup

In the default configuration, one valve position corresponds to the **P-A** and **B-T** flow paths maximally open and the **P-B** and **A-T** flow paths maximally closed (position **I** in the figure). Another valve position corresponds to the inverse configuration, with **P-B** and **A-T** maximally open and **P-A** and **B-T** maximally closed (position **II**). The third valve position corresponds to all flow paths maximally closed (position **III**). A translating spool serves as the valve control member and determines the position that the valve is in—**I**, **II**, **III**, or in between.

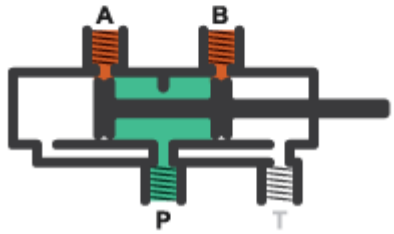
I.



II.



III.



Valve Positions

Physical signal port **S** controls the spool displacement. In the default configuration, a zero displacement signal corresponds to valve position **III**. A positive displacement signal shifts the spool toward valve position **I**. A negative displacement shifts the spool toward valve position **II**. The spool displacement acts indirectly by setting the spool position relative to each flow path—a length known here as the *orifice opening*. The orifice opening in turn determines the opening area of the respective flow path.

Orifice Openings

The orifice opening of a flow path depends partly on its *opening offset*—the orifice opening of a flow path at zero spool displacement. The block models only the effects of the opening offsets. An offset can be due to a change in distance between ports or spool lands—the thick disks built into the spool to obstruct flow. It can also be due to a change in the thicknesses of the spool lands. The orifice openings are computed separately for each flow path in terms of the respective opening offset:

$$h_{PA} = h_{PA0} + x$$

$$h_{PB} = h_{PB0} - x$$

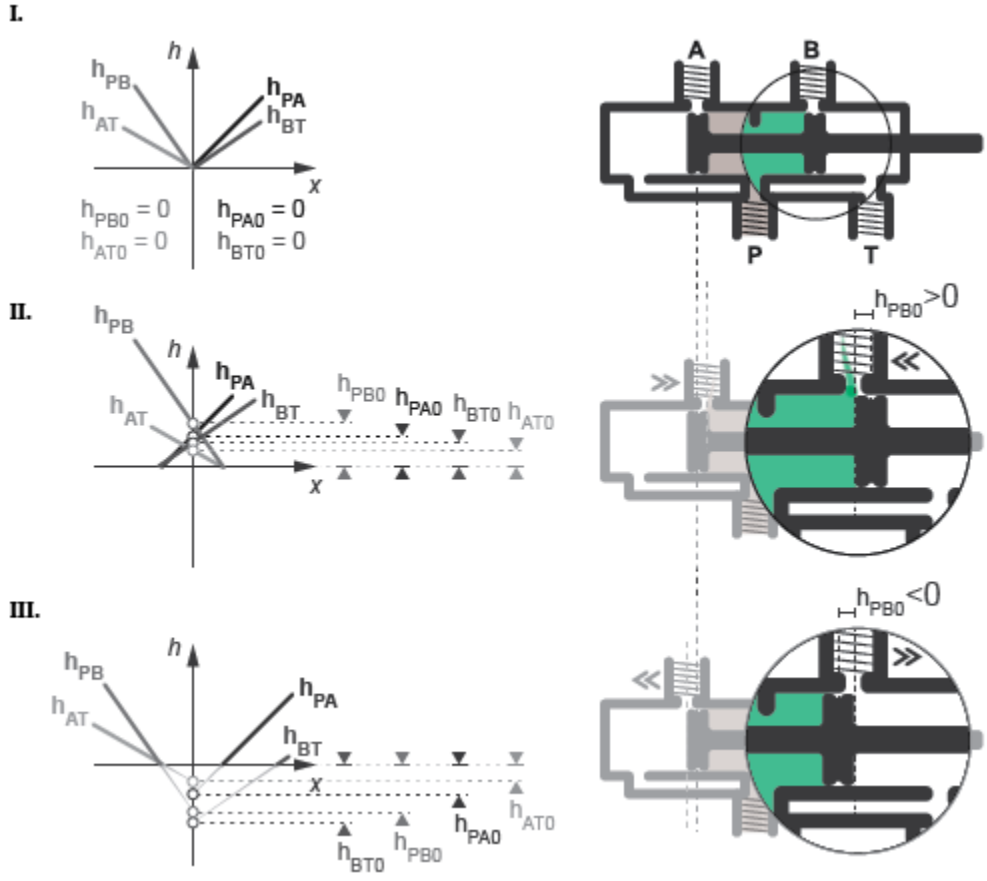
$$h_{AT} = h_{AT0} - x$$

$$h_{BT} = h_{BT0} + x$$

where:

- h_{PA}, h_{PB}, h_{AT}, and h_{BT} are the orifice openings of the **P**-**A**, **P-B**, **A**-**T**, and **P-B** flow paths. The orifice openings are computed during simulation.
- h_{PA0}, h_{PB0}, h_{AT0}, and h_{AT0} are the orifice opening offsets of the **P**-**A**, **P-B**, **A**-**T**, and **P-B** flow paths. The opening offsets are specified in the **Valve opening offsets** tab.
- x is the spool displacement relative to what in the zero-offset case is a fully closed valve. The spool displacement is specified through physical signal port **S**.

The figure shows the effects of the opening offsets on the orifice openings. Plot **I** corresponds to the default configuration with both opening offsets equal to zero. Plot **II** corresponds to a valve with both opening offsets greater than zero and plot **III** to a valve with both opening offsets smaller than zero. These cases are similar in behavior to zero-lapped (**I**), underlapped (**II**) and overlapped (**III**) valves. The valve schematics to the right show what the offset might look like. The circle highlights the offset in path **P-B**.



Zero (I), Positive (II), and Negative (III) Opening Offsets

An underlapped valve is always partially open and allows some flow at all spool displacements. An overlapped valve is fully closed over an extended range of spool displacements and requires longer spool travel to open. The table summarizes the opening offsets for zero-lapped, underlapped, and overlapped valves. Other configurations are possible—e.g., with one opening offset positive and the other negative.

Valve Lapping	Opening Offsets
Zero-lapped (default)	All zero

Valve Lapping	Opening Offsets
Underlapped	All positive
Overlapped	All negative

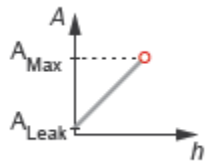
Opening Areas

The **Model parameterization** setting determines the calculations used for the opening areas of the flow paths—or, in the `Pressure-flow characteristic` case, the volumetric flow rates. The calculations are based on orifice parameters or tabulated data sets specified in the **Model Parameterization** tab. The block uses the same data for all flow paths if the **Area characteristics** parameter in the **Basic Parameters** tab is set to `Identical for all flow paths` and different data otherwise. Model parameterizations that you can select include:

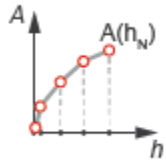
- `Maximum area and opening` — Specify the maximum opening area and the corresponding orifice opening. The opening area is a linear function of the orifice opening,

  $$A_{AT} = \frac{A_{AT,Max}}{h_{AT,Max}} h_{AT} + A_{Leak},$$

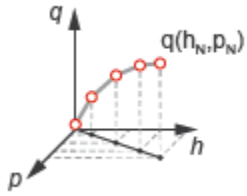
  where A is the opening area and h the orifice opening of a given flow path. The subscript `Max` refers to a fully open orifice and the subscript `Leak` to a fully closed orifice—one with internal leakage flow area only. The figure shows a plot of the linear function $A(h)$.

- `Area vs. opening table` — Specify the opening area at discrete orifice openings as a 1-D lookup table. The opening area is computed for a given orifice opening by interpolation or extrapolation of the tabulated data. The figure shows a conceptual plot of the tabulated function $A(h)$.



- `Pressure-flow characteristic` — Specify the volumetric flow rate at discrete orifice openings and pressure differentials as a 2-D lookup table. The opening area is computed for a given orifice opening and pressure differential by interpolation or extrapolation of the tabulated data. The figure shows a conceptual plot of the tabulated function $q(h, p)$.



Volumetric flow rates are computed analytically in the `Maximum area and opening` and `Area vs. opening table` parameterizations. The calculations are based on additional block parameters such as the flow discharge coefficient and account for the effects of flow regime—laminar or turbulent. Regime transition occurs at the specified critical laminar flow ratio or critical Reynolds number.

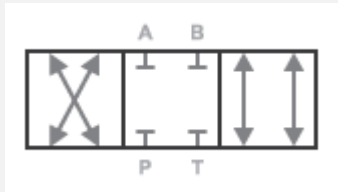
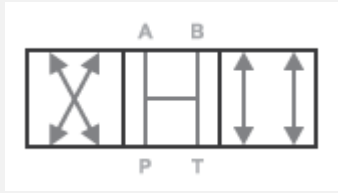
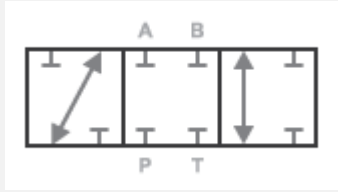
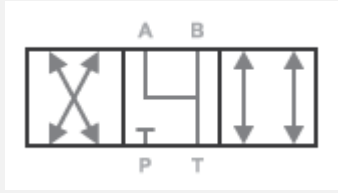
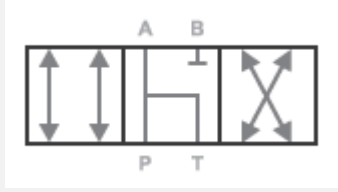
The `Maximum area and opening` and `Area vs. opening table` parameterizations also account for a small internal leakage area even in the fully closed state. The leakage area ensures that portions of the hydraulic network do not become isolated when a flow path is closed. Isolated, or “hanging”, network portions affect the computational efficiency of the model and can cause simulation to fail.

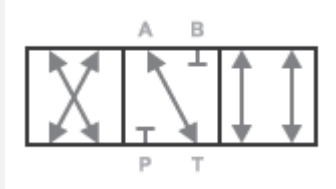
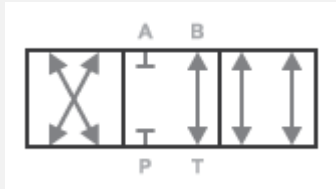
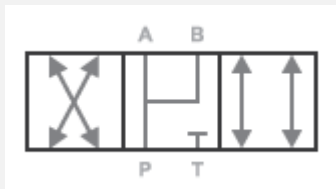
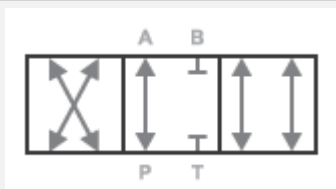
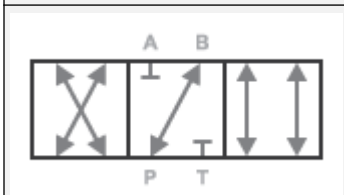
The effects of flow regime and internal leakage are assumed to be reflected in the tabulated flow rate data specified directly in the `Pressure-flow characteristic` parameterization.

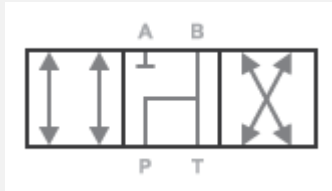
Valve Configurations

The opening offsets are by default zero. This configuration corresponds to a valve with all flow paths closed in the neutral position (**III** in the “Valve Positions” on page 1-120 schematic). Many other configurations exist. You can model a specific configuration by setting the opening offsets as shown in the table. All opening offset parameters are in the **Valve opening offsets** tab of the block dialog box

Basic 4-Way Directional Valve Configurations

Configuration	Valve opening offsets
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0
	• **Between ports P and A** > 0 • **Between ports P and B** > 0 • **Between ports A and T** > 0 • **Between ports B and T** > 0
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** < `-Maximum valve opening` • **Between ports B and T** < `-Maximum valve opening`
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** > 0 • **Between ports B and T** > 0
	• **Between ports P and A** > 0 • **Between ports P and B** < 0 • **Between ports A and T** > 0 • **Between ports B and T** < 0

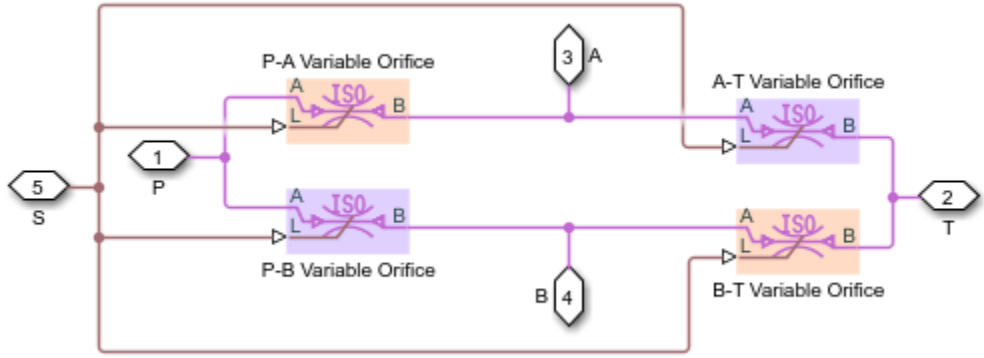
Configuration	Valve opening offsets
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** > 0 • **Between ports B and T** < 0
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** < 0 • **Between ports B and T** > 0
	• **Between ports P and A** > 0 • **Between ports P and B** > 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0
	• **Between ports P and A** > 0 • **Between ports P and B** < 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0
	• **Between ports P and A** < 0 • **Between ports P and B** > 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0

Configuration	Valve opening offsets
	• **Between ports P and A** < 0 • **Between ports P and B** > 0 • **Between ports A and T** < 0 • **Between ports B and T** > 0

Structural Component Diagram

The block is a composite component with four Variable Orifice blocks driven by a single physical signal. Blocks `Variable Orifice P-A` and `Variable Orifice P-B` represent the **P-A** and **P-B** flow paths. Blocks `Variable orifice A-T` and `Variable orifice B-T` represent the **A-T** and **B-T** flow paths. The physical signal is specified through Connection Port block **S**.

The **Orifice orientation** block parameters are set so that a positive signal acts to open `Variable Orifice P-A` and `Variable Orifice B-T` while closing `Variable Orifice A-T` and `Variable Orifice P-B`. A negative signal has the opposite effect—acting to open `Variable Orifice A-T` and `Variable Orifice P-B` while closing `Variable Orifice P-A` and `Variable Orifice B-T`.



Valve Structural Diagram

Assumptions

- Fluid inertia is ignored.
- Control member loading due to inertial, spring, and other forces is ignored.
- All valve orifices are assumed identical in size unless otherwise specified.

Ports

Input

S — Control member displacement, `m`
physical signal

Physical signal input port for the control member displacement.

Conserving

P — Pressure supply connection port
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port associated with the pressure supply line inlet.

T — Return line connection port
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port associated with the return line connection.

A — Actuator connection port
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port associated with the actuator connection port.

B — Actuator connection port
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port associated with the actuator connection port.

Parameters

Basic Parameters

`Area characteristics` — Symmetry of the flow path opening characteristics
`Identical for all flow paths` (default) | `Different for each flow path`

Choice of different or identical flow path opening characteristics. Select `Different for each flow path` to specify flow path parameters or tabulated data separately for each flow path.

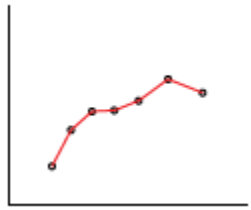
`Model parameterization` — Types of valve parameters to use as input
`Maximum area and opening` (default) | `Area vs. opening table` | `Pressure-flow characteristic`

Parameterization of the valve model. Options include:

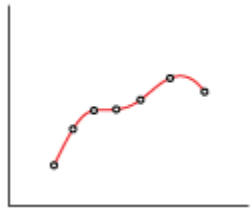
- `By maximum area and opening` — Specify the maximum orifice opening and opening area. The opening area varies linearly with the spool displacement specified at physical signal port **S**.
- `By area vs. opening table` — Specify the flow path opening area at discrete orifice openings as a 1-D lookup table. The opening area is computed by interpolation or extrapolation of the tabulated data.
- `By pressure-flow characteristic` — Specify the flow path volumetric flow rates at discrete orifice openings and pressure differentials. The flow rate is computed by interpolation or extrapolation of the tabulated data.

`Interpolation method` — Method of computing values inside the lookup table range
`Linear` (default) | `Smooth`

Method of computing values inside the tabulated data range. The `Linear` method joins adjacent data points with straight line or surface segments with generally discontinuous slope at the segment boundaries. Surface segments are used in the 2-D lookup table specified in the `Pressure-flow characteristic` model parameterization.



The `Smooth` method replaces the straight segments with curved versions that have continuous slope everywhere inside the tabulated data range. The segments form a smooth line or surface passing through all of the tabulated data points without the discontinuities in first-order derivatives characteristic of the `Linear` interpolation method.



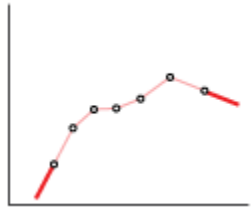
Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By area vs. opening table` or `Pressure-flow characteristic`.

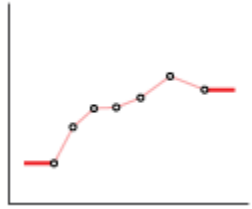
`Extrapolation method` — Method of computing values outside of the lookup table range

`Linear` (default) | `Nearest`

Method of computing values outside of the tabulated data range. The `Linear` method extends the line segment drawn between the last two data points at each end of the data range outward with a constant slope.



The `Nearest` method extends the last data point at each end of the data range outward as a horizontal line with constant value.



Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By area vs. opening table` or `Pressure-flow characteristic`.

`Flow discharge coefficient` — Semi-empirical parameter for valve capacity characterization

`0.7` (default) | unitless scalar between 0 and 1

Ratio of the actual and theoretical flow rates through the valve. This parameter depends on the geometrical properties of the valve. Values are usually provided in textbooks and manufacturer data sheets.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table`.

`Leakage area` — Internal leakage area between valve inlets

`1e-12 m^2` (default) | positive scalar in units of area

Total area of internal leaks in the completely closed state. The purpose of this parameter is to maintain the numerical integrity of the fluid network by preventing a portion of that network from becoming isolated when the valve is completely closed.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table`.

`Laminar transition specification` — Choice of parameter used to define the laminar-turbulent transition

`Pressure ratio` (default) | `Reynolds number`

Select the parameter to base the laminar-turbulent transition on. Options include:

- `Pressure ratio` — Flow transitions between laminar and turbulent at the pressure ratio specified in the **Laminar flow pressure ratio** parameter. Use this option for the smoothest and most numerically robust flow transitions.
- `Reynolds number` — The transition occurs at the Reynolds number specified in the **Critical Reynolds number** parameter. Flow transitions are more abrupt and can cause simulation issues at near-zero flow rates.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table`.

`Laminar flow pressure ratio` — Pressure ratio at the laminar-turbulent flow transition

`0.999` (default) | unitless scalar between 0 and 1

Pressure ratio at which the flow transitions between the laminar and turbulent regimes. The pressure ratio is the fraction of the outlet pressure over the inlet pressure.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table` and the **Laminar transition specification** parameter is set to `Reynolds number`.

`Critical Reynolds number` — Reynolds number at the laminar-turbulent flow transition

12 (default) | unitless positive scalar

Maximum Reynolds number for laminar flow. This parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value, 12, corresponds to a round orifice in thin material with sharp edges.

Dependencies

This parameter is active when the **Model Parameterization** parameter is set to `By maximum area and opening` or `By area vs. opening table` and the **Laminar transition specification** parameter is set to `Reynolds number`.

Model Parameterization: Maximum area and opening

`Maximum opening` — Orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Orifice opening at which the opening area of a flow path is a maximum. The maximum opening is the same for all flow paths when this parameter is exposed. For the orifice opening definitions, see “Orifice Openings” on page 1-121.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`Maximum opening area` — Maximum cross-sectional area of a flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of a flow path. The maximum opening areas are the same for all paths when this parameter is exposed. For the opening area calculations, see “Opening Areas” on page 1-123.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`P-A, maximum opening` — `P-A` orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Control member position at which the P-A opening area is equal to the **P-A, maximum opening area** parameter value.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`P-A, maximum opening area` — Maximum flow area of the P-A flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of the **P-A** flow path. The maximum opening areas are different for each flow path when this parameter is exposed. For the opening area calculations, see “Opening Areas” on page 1-123.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`P-B, maximum opening` — `P-B` Orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Control member position at which the P-B opening area is equal to the **P-B, maximum opening area** parameter value.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`P-B, maximum opening area` — Maximum flow area of the P-B flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of the **P-B** flow path. The maximum opening areas are different for each flow path when this parameter is exposed. For the opening area calculations, see “Opening Areas” on page 1-123.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`A-T, maximum opening` — `A-T` Orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Control member position at which the A-T opening area is equal to the **A-T, maximum opening area** parameter value.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`A-T, maximum opening area` — Maximum flow area of the A-T flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of the **A-T** flow path. The maximum opening areas are different for each flow path when this parameter is exposed. For the opening area calculations, see “Opening Areas” on page 1-123.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`B-T, maximum opening` — `B-T` Orifice opening at which the opening area is a maximum

`0.005 m` (default) | positive scalar with units of length

Control member position at which the B-T opening area is equal to the **B-T, maximum opening area** parameter value.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

`B-T, maximum opening area` — Maximum flow area of the B-T flow path

`0.5e-4 m^2` (default) | positive scalar with units of area

Maximum cross-sectional area of the **B-T** flow path. The maximum opening areas are different for each flow path when this parameter is exposed. For the opening area calculations, see “Opening Areas” on page 1-123.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Maximum area and opening`.

Model Parameterization: Area vs. opening table

`Opening vector` — Orifice openings at which to specify the flow path opening areas

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of orifice openings used to construct the opening area 1-D lookup table for all flow paths. The opening characteristics are identical for all flow paths when this parameter is exposed. The number of elements in the array determines the number of elements in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`Opening area vector` — Opening areas corresponding to the orifice openings specified in the `Opening vector` parameter

`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of flow path opening areas corresponding to the specified orifice openings. The opening characteristics are identical for all flow paths when this parameter is exposed.

The number of elements in the array must match the number of elements in the **Opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`P-A, opening vector` — `P-A` orifice openings at which to specify the `P-A` opening area

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **P-A** orifice openings used to construct the opening area 1-D lookup table for the **P-A** flow path. The number of elements in the array determines the number of elements in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`P-A, opening area vector` — `P-A` opening areas corresponding to the orifice openings specified in the `P-A, opening vector` parameter

`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of **P-A** opening areas corresponding to the specified **P-A** orifice openings. The number of elements in the array must match the number of elements in the **P-A, opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`P-B, opening vector` — `P-B` orifice openings at which to specify the `P-B` opening area

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **P-B** orifice openings used to construct the opening area 1-D lookup table for the **P-B** flow path. The number of elements in the array determines the number of elements in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`P-B, opening area vector` — `P-B` opening areas corresponding to the orifice openings specified in the `P-B, opening vector` parameter

`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of **P-B** opening areas corresponding to the specified **P-B** orifice openings. The number of elements in the array must match the number of elements in the **P-B, opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`A-T, opening vector` — `A-T` orifice openings at which to specify the `A-T` opening area

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **A-T** orifice openings used to construct the opening area 1-D lookup table for the **A-T** flow path. The number of elements in the array determines the number of elements in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`A-T, opening area vector` — `A-T` opening areas corresponding to the orifice openings specified in the `A-T, opening vector` parameter

`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of **A-T** opening areas corresponding to the specified **A-T** orifice openings. The number of elements in the array must match the number of elements in the **A-T, opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`B-T, opening vector` — `B-T` orifice openings at which to specify the `B-T` opening area

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **B-T** orifice openings used to construct the opening area 1-D lookup table for the **B-T** flow path. The number of elements in the array determines the number of elements in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

`B-T, opening area vector` — `B-T` opening areas corresponding to the orifice openings specified in the `B-T, opening vector` parameter

`[1.0e-9, 2.0352e-7, 4.0736e-5, 0.00011438, 0.00034356] m^2` (default) | numerical array with units of area

Array of **B-T** opening areas corresponding to the specified **B-T** orifice openings. The number of elements in the array must match the number of elements in the **B-T, opening vector** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Area vs. opening table`.

Model Parameterization: Pressure-flow characteristic

`Opening vector, s` — Orifice openings at which to specify the flow rate tabulated data

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of orifice openings used to construct the volumetric flow rate 2-D lookup table. The number of elements in the array determines the number of rows in the table. The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`Pressure differential vector, dp` — Pressure differentials at which to specify the flow rate data

`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table. The number of elements in the array determines the number of columns in the table. The array must increase from left to right but the intervals between the array

element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`Volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified control member displacements and pressure differentials

5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates corresponding to the specified orifice openings and pressure differentials. The number of rows must match the number of elements in the **Opening vector, s** parameter. The number of columns must match the number of elements in the **Pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Identical for all flow paths` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-A, opening vector, s` — Orifice openings at which to specify the `P-A` flow rate data

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of **P-A** orifice openings used to construct the **P-A** volumetric flow rate 2-D lookup table. The number of elements in the array determines the number of rows in the table. The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-A, pressure differential vector, dp` — Pressure differentials at which to specify the flow rate data for the P-A flow path
`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table for the P-A flow path. The number of elements in the array determines the number of columns in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-A, volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified control member displacements and pressure differentials for the P-A flow path
5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates through the P-A flow path at the specified control member displacements and pressure differentials. The number of rows must match the number of elements in the **P-A, opening vector, s** parameter. The number of columns must match the number of elements in the **P-A, pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-B, opening vector, s` — Control member displacements at which to specify the flow rate data for the P-B flow path
`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of control member displacements used to construct the volumetric flow rate 2-D lookup table for the P-B flow path. The number of elements in the array determines the number of rows in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-B, pressure differential vector, dp` — Pressure differentials at which to specify the flow rate data for the P-B flow path

`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table for the P-B flow path. The number of elements in the array determines the number of columns in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`P-B, volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified control member displacements and pressure differentials for the P-B flow path

5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates through the P-B flow path at the specified control member displacements and pressure differentials. The number of rows must match the number of elements in the **P-B, opening vector, s** parameter. The number of columns

must match the number of elements in the **P-A, pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`A-T, opening vector, s` — Control member displacements at which to specify the flow rate data for the A-T flow path

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of control member displacements used to construct the volumetric flow rate 2-D lookup table for the A-T flow path. The number of elements in the array determines the number of rows in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`A-T, pressure differential vector, dp` — Pressure differentials at which to specify the flow rate data for the A-T flow path

`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table for the A-T flow path. The number of elements in the array determines the number of columns in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`A-T, volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified control member displacements and pressure differentials for the A-T flow path

5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates through the A-T flow path at the specified control member displacements and pressure differentials. The number of rows must match the number of elements in the **A-T, opening vector, s** parameter. The number of columns must match the number of elements in the **A-T, pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`B-T, opening vector, s` — Control member displacements at which to specify the flow rate data for the B-T flow path

`[-0.002, 0.0, 0.002, 0.005, 0.015] m` (default) | numerical array with units of length

Array of control member displacements used to construct the volumetric flow rate 2-D lookup table for the B-T flow path. The number of elements in the array determines the number of rows in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`B-T, pressure differential vector, dp` — Pressure differentials at which to specify the flow rate data for the B-T flow path
`[-10000000.0, -5000000.0, -2000000.0, 2000000.0, 5000000.0, 10000000.0] Pa` (default) | numerical array with units of pressure

Array of pressure differentials used to construct the volumetric flow rate 2-D lookup table for the B-T flow path. The number of elements in the array determines the number of columns in the table.

The array must increase from left to right but the intervals between the array element values need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

`B-T, volumetric flow rate table, q(s,dp)` — Volumetric flow rates at the specified control member displacements and pressure differentials for the B-T flow path
5-by-6 matrix with elements between `-0.0344` and `0.0344` `m^3/s` (default) | numerical array with units of length

Matrix with the volumetric flow rates through the B-T flow path at the specified control member displacements and pressure differentials. The number of rows must match the number of elements in the **B-T, opening vector, s** parameter. The number of columns must match the number of elements in the **B-T, pressure differential vector, dp** parameter.

Dependencies

This parameter is active when the **Area characteristics** parameter is set to `Different for each flow path` and the **Model parameterization** parameter is set to `Pressure-flow characteristic`.

Valve Opening Offsets

`Between P-A` — `P-A` orifice opening at zero spool displacement
`0 m` (default) | scalar number with units of lengths

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve.

`Between P-B` — `P-B` orifice opening at zero spool displacement
`0 m` (default) | scalar number with units of lengths

Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve.

`Between A-T` — `A-T` orifice opening at zero spool displacement
`0 m` (default) | scalar number with units of length

Orifice opening of the **A-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve.

`Between B-T` — `BT` orifice opening at zero spool displacement
`0 m` (default) | scalar number with units of length

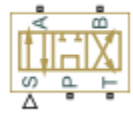
Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve.

See Also

2-Way Directional Valve | 3-Way Directional Valve

4-Way Directional Valve A

Configuration A of hydraulic continuous 4-way directional valve

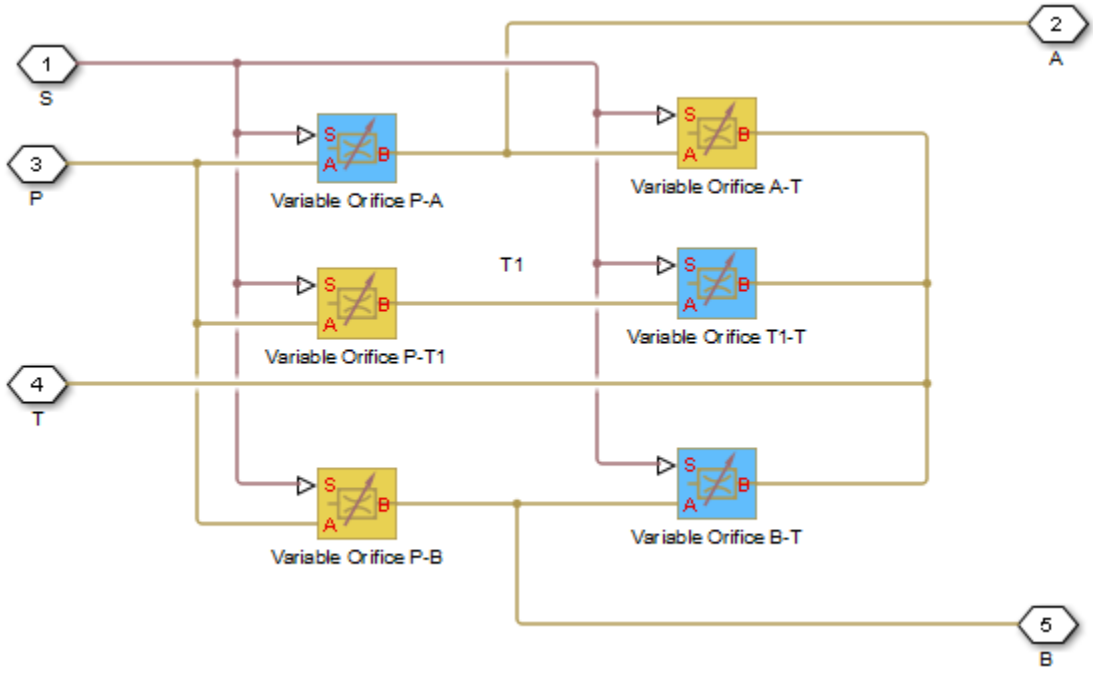


Library

Directional Valves

Description

The 4-Way Directional Valve A block simulates a configuration of hydraulic continuous 4-way directional valve where the valve unloads the pump in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



All blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A, B-T, and T1-T) and closes the orifices colored yellow (orifices A-T, P-B, and P-T1). As a result, the openings of the orifices are computed as follows:

$$h_{PA} = h_{PA0} + x$$

$$h_{PB} = h_{PB0} - x$$

$$h_{AT} = h_{AT0} - x$$

$$h_{BT} = h_{BT0} + x$$

$$h_{PT1} = h_{PT10} - x$$

$$h_{T1T} = h_{T1T0} + x$$

where

h_{PA}	Orifice opening for the `Variable Orifice P-A` block
h_{PB}	Orifice opening for the `Variable Orifice P-B` block
h_{AT}	Orifice opening for the `Variable Orifice A-T` block
h_{BT}	Orifice opening for the `Variable Orifice B-T` block
h_{PT1}	Orifice opening for the `Variable Orifice P-T1` block
h_{T1T}	Orifice opening for the `Variable Orifice T1-T` block
h_{PA0}	Initial opening for the `Variable Orifice P-A` block
h_{PB0}	Initial opening for the `Variable Orifice P-B` block
h_{AT0}	Initial opening for the `Variable Orifice A-T` block
h_{BT0}	Initial opening for the `Variable Orifice B-T` block
h_{PT10}	Initial opening for the `Variable Orifice P-T1` block
h_{T1T0}	Initial opening for the `Variable Orifice T1-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-151
- “Valve Opening Offsets Tab” on page 1-155

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly

dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e +06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.

- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and A

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and B

Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports A and T

Orifice opening of the **A-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports B and T

Orifice opening of the **B-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and T1

Orifice opening of the **P-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports T1 and T

Orifice opening of the **T1–T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

See Also

4-Way Directional Valve D | 4-Way Directional Valve K | 4-Way Directional Valve E | 4-Way Directional Valve | 4-Way Directional Valve B | 4-Way Directional Valve C | 4-Way Directional Valve F | 4-Way Directional Valve G | 4-Way Directional Valve H | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve B

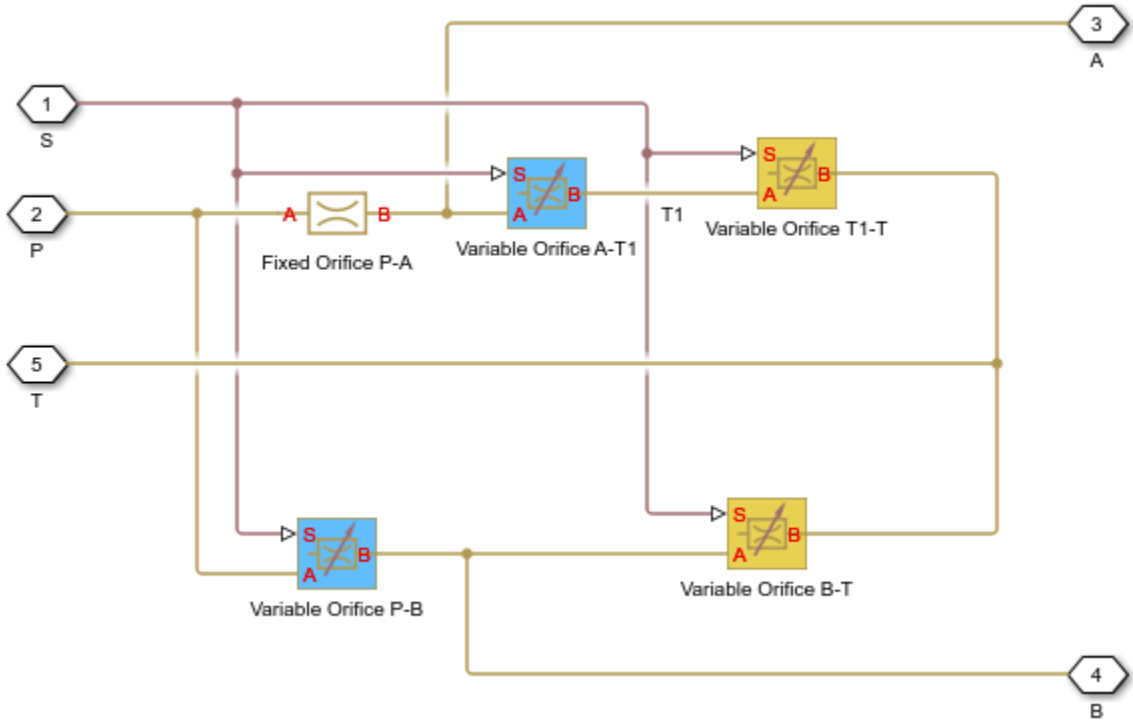
Configuration B of hydraulic continuous 4-way directional valve

Library

Directional Valves

Description

The 4-Way Directional Valve B block simulates a configuration of hydraulic continuous 4-way directional valve where port A is permanently connected to port P, and all four ports are interconnected in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of a Fixed Orifice block in the P-A path and four Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices A-T1 and P-B) and closes the orifices colored yellow (orifices T1-T and B-T). As a result, the openings of the orifices are computed as follows:

$$h_{AT1} = h_{AT10} - x$$

$$h_{PB} = h_{PB0} - x$$

$$h_{T1T} = h_{T1T0} + x$$

$$h_{BT} = h_{BT0} + x$$

where

h_{AT1}	Orifice opening for the `Variable Orifice A-T1` block
h_{PB}	Orifice opening for the `Variable Orifice P-B` block
h_{T1T}	Orifice opening for the `Variable Orifice T1-T` block
h_{BT}	Orifice opening for the `Variable Orifice B-T` block
h_{AT10}	Initial opening for the `Variable Orifice A-T1` block
h_{PB0}	Initial opening for the `Variable Orifice P-B` block
h_{T1T0}	Initial opening for the `Variable Orifice T1-T` block
h_{BT0}	Initial opening for the `Variable Orifice B-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-159
- “Valve Opening Offsets Tab” on page 1-163

Basic Parameters Tab

Compared to the Basic Parameters tab of the 4-Way Directional Valve block, there is one additional parameter:

Path P-A passage area

Specify the area of the P-A passage. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

The rest of the parameters on this tab are identical to those of the 4-Way Directional Valve block:

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector.

All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and B

Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports B and T

Orifice opening of the **B-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports A and T1

Orifice opening of the **A-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports T1 and T

Orifice opening of the **T1–T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pressure supply line inlet.

`T`

Hydraulic conserving port associated with the return line connection.

`A`

Hydraulic conserving port associated with the actuator connection port.

`B`

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

See Also

4-Way Directional Valve D | 4-Way Directional Valve K | 4-Way Directional Valve E | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve C | 4-Way Directional Valve F | 4-Way Directional Valve G | 4-Way Directional Valve H | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve C

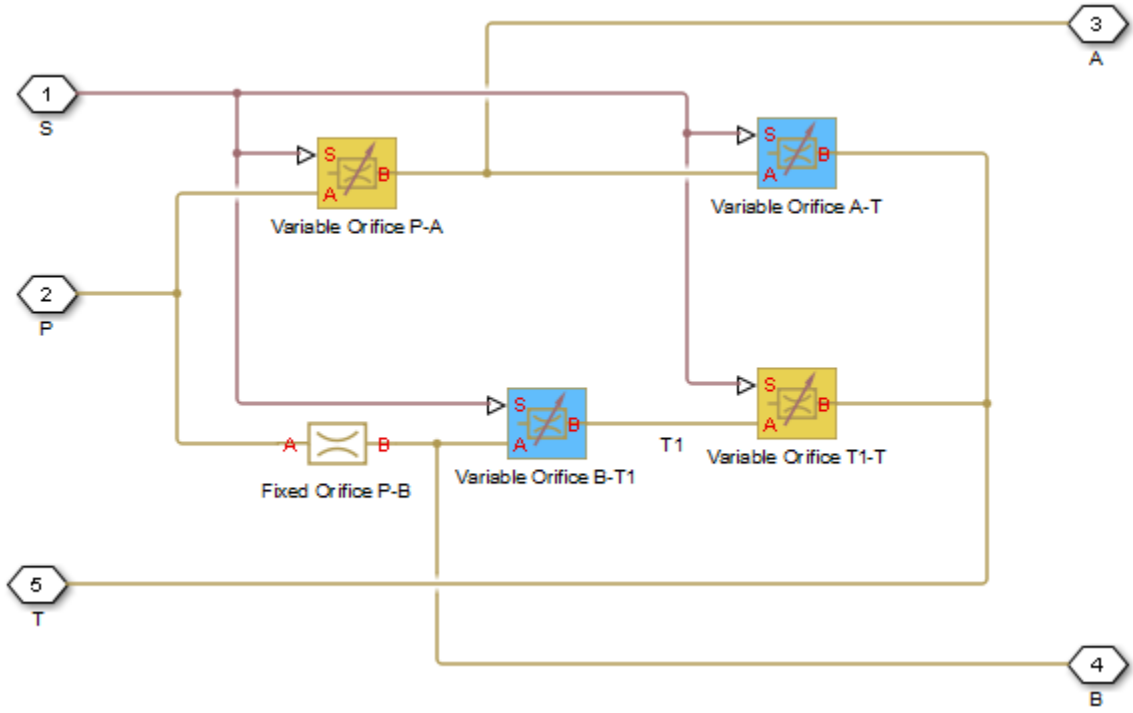
Configuration C of hydraulic continuous 4-way directional valve

Library

Directional Valves

Description

The 4-Way Directional Valve C block simulates a configuration of hydraulic continuous 4-way directional valve where port P is connected to port A and port B is connected to port T in the left position; in the right position, both port A and port B are connected to P; and all ports are interconnected in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of a Fixed Orifice block in the P-B path and four Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices B-T1 and A-T) and closes the orifices colored yellow (orifices P-A and T1-T). As a result, the openings of the orifices are computed as follows:

$$h_{PA} = h_{PA0} - x$$

$$h_{AT} = h_{AT0} + x$$

$$h_{BT1} = h_{BT10} + x$$

$$h_{T1T} = h_{T1T0} - x$$

where

h_{PA}	Orifice opening for the `Variable Orifice P-A` block
h_{AT}	Orifice opening for the `Variable Orifice A-T` block
h_{BT1}	Orifice opening for the `Variable Orifice B-T1` block
h_{T1T}	Orifice opening for the `Variable Orifice T1-T` block
h_{PA0}	Initial opening for the `Variable Orifice P-A` block
h_{PB0}	Initial opening for the `Variable Orifice P-B` block
h_{BT10}	Initial opening for the `Variable Orifice B-T1` block
h_{T1T0}	Initial opening for the `Variable Orifice T1-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-167
- “Valve Opening Offsets Tab” on page 1-171

Basic Parameters Tab

Compared to the Basic Parameters tab of the 4-Way Directional Valve block, there is one additional parameter:

Path P-B passage area

Specify the area of the P-B passage. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

The rest of the parameters on this tab are identical to those of the 4-Way Directional Valve block:

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector.

All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and A

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports A and T

Orifice opening of the **A-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports B and T1

Orifice opening of the **B-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports T1 and T

Orifice opening of the **T1–T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pressure supply line inlet.

`T`

Hydraulic conserving port associated with the return line connection.

`A`

Hydraulic conserving port associated with the actuator connection port.

`B`

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

See Also

4-Way Directional Valve D | 4-Way Directional Valve K | 4-Way Directional Valve E | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way Directional Valve F | 4-Way Directional Valve G | 4-Way Directional Valve H | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve D

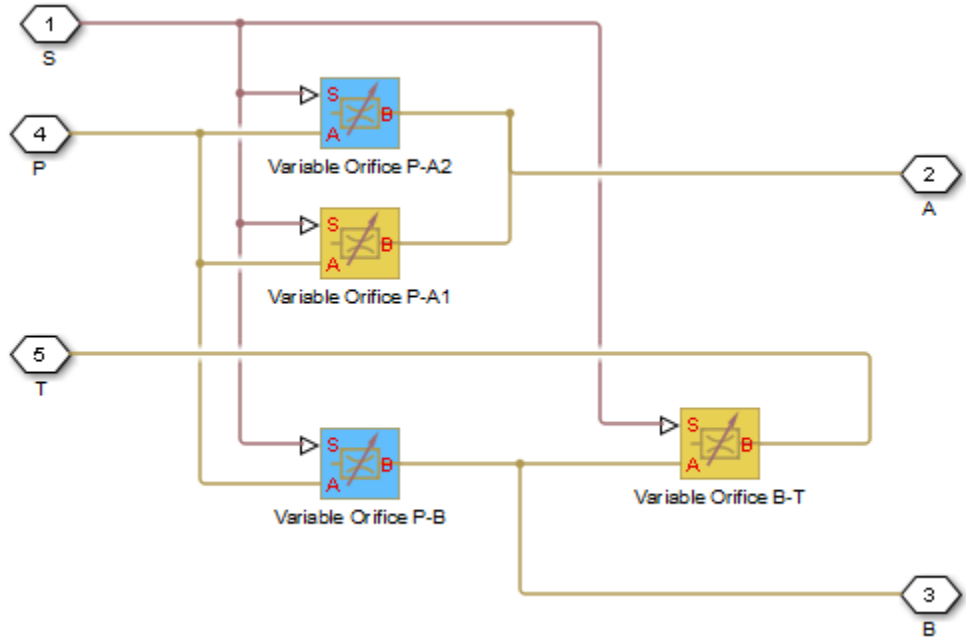
Configuration D of hydraulic continuous 4-way directional valve

Library

Directional Valves

Description

The 4-Way Directional Valve D block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the left valve position. In the right position, port P is connected to port A, while port B is connected to port T. All connections are blocked in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of four Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A2 and P-B) and closes the orifices colored yellow (orifices P-A1 and B-T). As a result, the openings of the orifices are computed as follows:

$$h_{PA1} = h_{PA10} - x$$

$$h_{PA2} = h_{PA20} + x$$

$$h_{PB} = h_{PB0} + x$$

$$h_{BT} = h_{BT0} - x$$

where

h_{PA1}	Orifice opening for the `Variable Orifice P-A1` block
h_{PA2}	Orifice opening for the `Variable Orifice P-A2` block
h_{PB}	Orifice opening for the `Variable Orifice P-B` block
h_{BT}	Orifice opening for the `Variable Orifice B-T` block
h_{PA10}	Initial opening for the `Variable Orifice P-A1` block
h_{PA20}	Initial opening for the `Variable Orifice P-A2` block
h_{PB0}	Initial opening for the `Variable Orifice P-B` block
h_{BT0}	Initial opening for the `Variable Orifice B-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-175
- “Valve Opening Offsets Tab” on page 1-178

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is

determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and A1

Orifice opening of the **P-A1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and A2

Orifice opening of the **P-A2** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and B

Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports B and T

Orifice opening of the **B-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pressure supply line inlet.

`T`

Hydraulic conserving port associated with the return line connection.

`A`

Hydraulic conserving port associated with the actuator connection port.

`B`

Hydraulic conserving port associated with the actuator connection port.

`S`

Physical signal port to control spool displacement.

See Also

4-Way Directional Valve C | 4-Way Directional Valve K | 4-Way Directional Valve E | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way

Directional Valve F | 4-Way Directional Valve G | 4-Way Directional Valve H | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve E

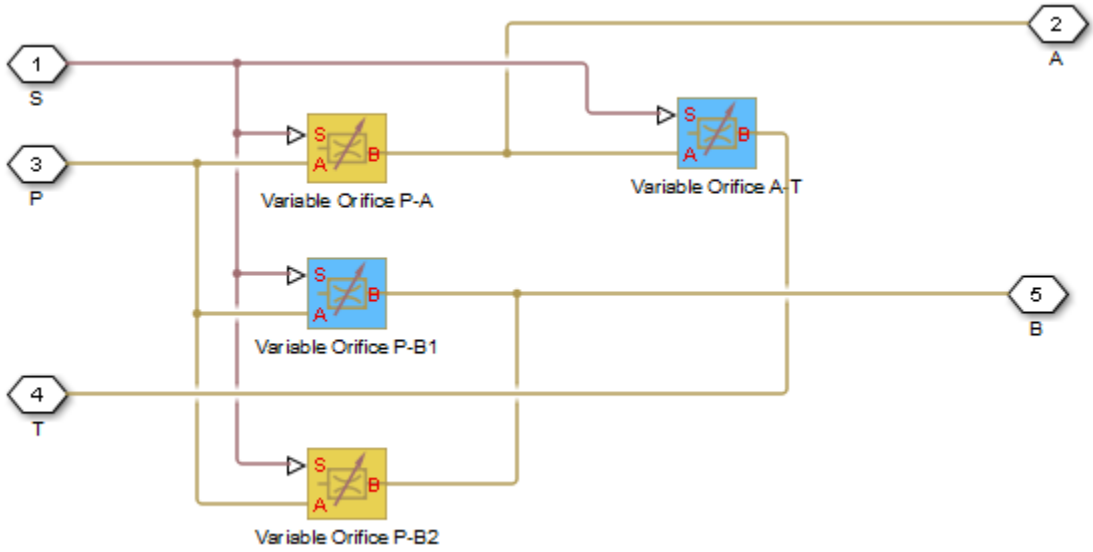
Configuration E of hydraulic continuous 4-way directional valve

Library

Directional Valves

Description

The 4-Way Directional Valve E block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the right valve position. In the left position, port P is connected to port B, while port A is connected to port T. All connections are blocked in neutral position. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of four Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-B1 and A-T) and closes the orifices colored yellow (orifices P-A and P-B2). As a result, the openings of the orifices are computed as follows:

$$h_{PA} = h_{PA0} - x$$

$$h_{PB1} = h_{PB10} + x$$

$$h_{PB2} = h_{PB20} - x$$

$$h_{AT} = h_{AT0} + x$$

where

h_{PA}	Orifice opening for the `Variable Orifice P-A` block
h_{PB1}	Orifice opening for the `Variable Orifice P-B1` block

h_{PB2}	Orifice opening for the `Variable Orifice P-B2` block
h_{AT}	Orifice opening for the `Variable Orifice A-T` block
h_{PA0}	Initial opening for the `Variable Orifice P-A` block
h_{PB10}	Initial opening for the `Variable Orifice P-B1` block
h_{PB20}	Initial opening for the `Variable Orifice P-B2` block
h_{AT0}	Initial opening for the `Variable Orifice A-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-183
- “Valve Opening Offsets Tab” on page 1-186

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e +06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow

rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and A

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports A and T

Orifice opening of the **A-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and B1

Orifice opening of the **P-B1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and B2

Orifice opening of the **P-B2** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pressure supply line inlet.

`T`

Hydraulic conserving port associated with the return line connection.

`A`

Hydraulic conserving port associated with the actuator connection port.

`B`

Hydraulic conserving port associated with the actuator connection port.

`S`

Physical signal port to control spool displacement.

See Also

4-Way Directional Valve C | 4-Way Directional Valve K | 4-Way Directional Valve D | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way Directional Valve F | 4-Way Directional Valve G | 4-Way Directional Valve H | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve F

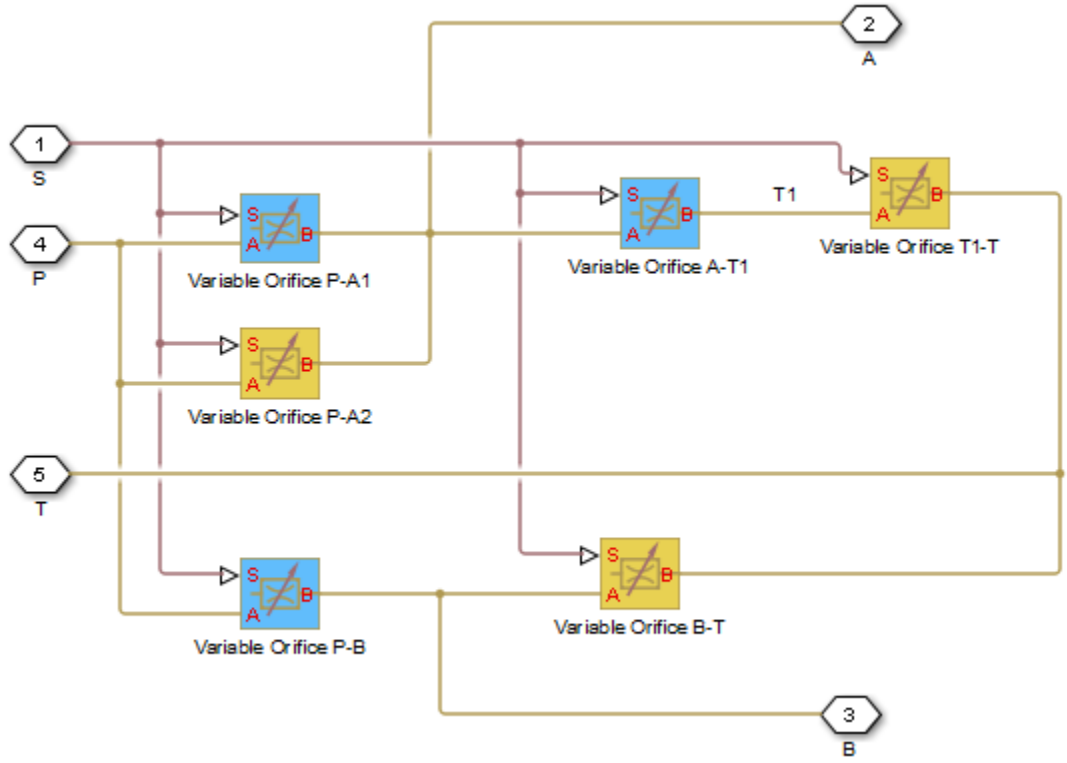
Configuration F of hydraulic continuous 4-way directional valve

Library

Directional Valves

Description

The 4-Way Directional Valve F block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the left valve position. In the right position, port P is connected to port A, while port B is connected to port T. In neutral position, ports A and B are connected to port T. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A1, A-T1, and P-B) and closes the orifices colored yellow (orifices P-A2, B-T, and T1-T). As a result, the openings of the orifices are computed as follows:

$$h_{PA1} = h_{PA10} + x$$

$$h_{PA2} = h_{PA20} - x$$

$$h_{AT1} = h_{AT10} + x$$

$$h_{T1T} = h_{T1T0} - x$$

$$h_{PB} = h_{PB0} + x$$

$$h_{BT} = h_{BT0} - x$$

where

h_{PA1}	Orifice opening for the `Variable Orifice P-A1` block
h_{PA2}	Orifice opening for the `Variable Orifice P-A2` block
h_{AT1}	Orifice opening for the `Variable Orifice A-T1` block
h_{T1T}	Orifice opening for the `Variable Orifice T1-T` block
h_{PB}	Orifice opening for the `Variable Orifice P-B` block
h_{BT}	Orifice opening for the `Variable Orifice B-T` block
h_{PA10}	Initial opening for the `Variable Orifice P-A1` block
h_{PA20}	Initial opening for the `Variable Orifice P-A2` block
h_{AT10}	Initial opening for the `Variable Orifice A-T1` block
h_{T1T0}	Initial opening for the `Variable Orifice T1-T` block
h_{PB0}	Initial opening for the `Variable Orifice P-B` block
h_{BT0}	Initial opening for the `Variable Orifice B-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-191
- “Valve Opening Offsets Tab” on page 1-194

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings**

values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and A1

Orifice opening of the **P-A1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and A2

Orifice opening of the **P-A2** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports A and T1

Orifice opening of the **A-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports T1 and T

Orifice opening of the **T1-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports P and B

Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports B and T

Orifice opening of the **B-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped

valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pressure supply line inlet.

`T`

Hydraulic conserving port associated with the return line connection.

`A`

Hydraulic conserving port associated with the actuator connection port.

`B`

Hydraulic conserving port associated with the actuator connection port.

`S`

Physical signal port to control spool displacement.

See Also

4-Way Directional Valve C | 4-Way Directional Valve K | 4-Way Directional Valve D | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way Directional Valve E | 4-Way Directional Valve G | 4-Way Directional Valve H | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve G

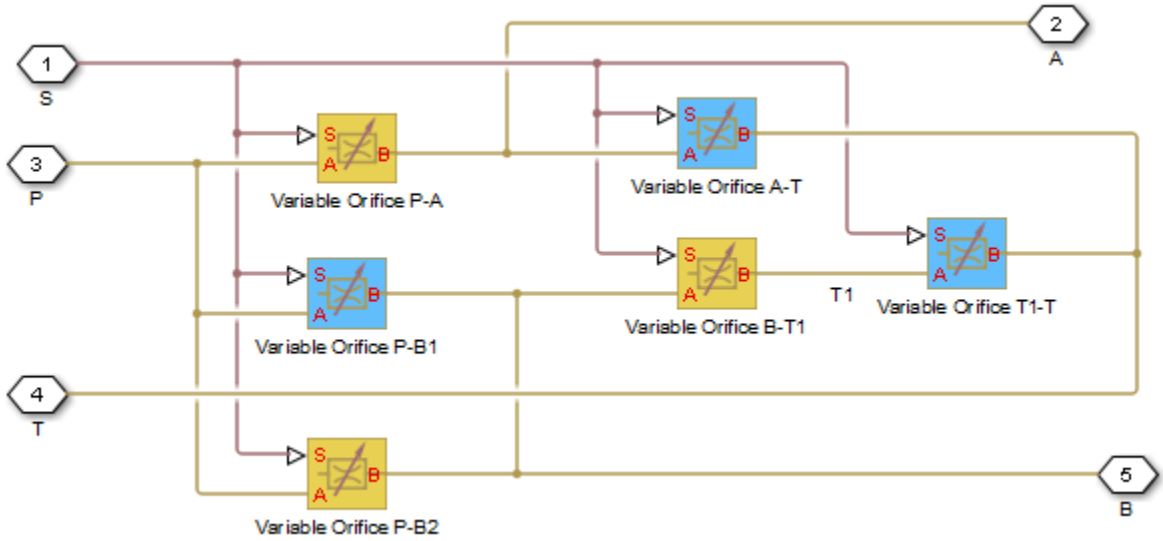
Configuration G of hydraulic continuous 4-way directional valve

Library

Directional Valves

Description

The 4-Way Directional Valve G block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the right valve position. In the left position, port P is connected to port B, while port A is connected to port T. In neutral position, ports A and B are connected to port T. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-B1, A-T, and T1-T) and closes the orifices colored yellow (orifices P-A, B-T1, and P-B2). As a result, the openings of the orifices are computed as follows:

$$h_{PB1} = h_{PB10} + x$$

$$h_{PB2} = h_{PB20} - x$$

$$h_{BT1} = h_{BT10} - x$$

$$h_{T1T} = h_{T1T0} + x$$

$$h_{PA} = h_{PA0} - x$$

$$h_{AT} = h_{AT0} + x$$

where

h_{PB1}	Orifice opening for the `Variable Orifice P-B1` block
h_{PB2}	Orifice opening for the `Variable Orifice P-B2` block
h_{BT1}	Orifice opening for the `Variable Orifice B-T1` block
h_{T1T}	Orifice opening for the `Variable Orifice T1-T` block
h_{PA}	Orifice opening for the `Variable Orifice P-A` block
h_{AT}	Orifice opening for the `Variable Orifice A-T` block
h_{PB10}	Initial opening for the `Variable Orifice P-B1` block
h_{PB20}	Initial opening for the `Variable Orifice P-B2` block
h_{BT10}	Initial opening for the `Variable Orifice B-T1` block
h_{T1T0}	Initial opening for the `Variable Orifice T1-T` block
h_{PA0}	Initial opening for the `Variable Orifice P-A` block
h_{AT0}	Initial opening for the `Variable Orifice A-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-198
- “Valve Opening Offsets Tab” on page 1-201

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced.

The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and B1

Orifice opening of the **P-B1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped

valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and B2

Orifice opening of the **P-B2** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports B and T1

Orifice opening of the **B-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports T1 and T

Orifice opening of the **T1-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports P and A

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports A and T

Orifice opening of the **B-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

See Also

4-Way Directional Valve C | 4-Way Directional Valve K | 4-Way Directional Valve D | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way Directional Valve E | 4-Way Directional Valve F | 4-Way Directional Valve H | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve H

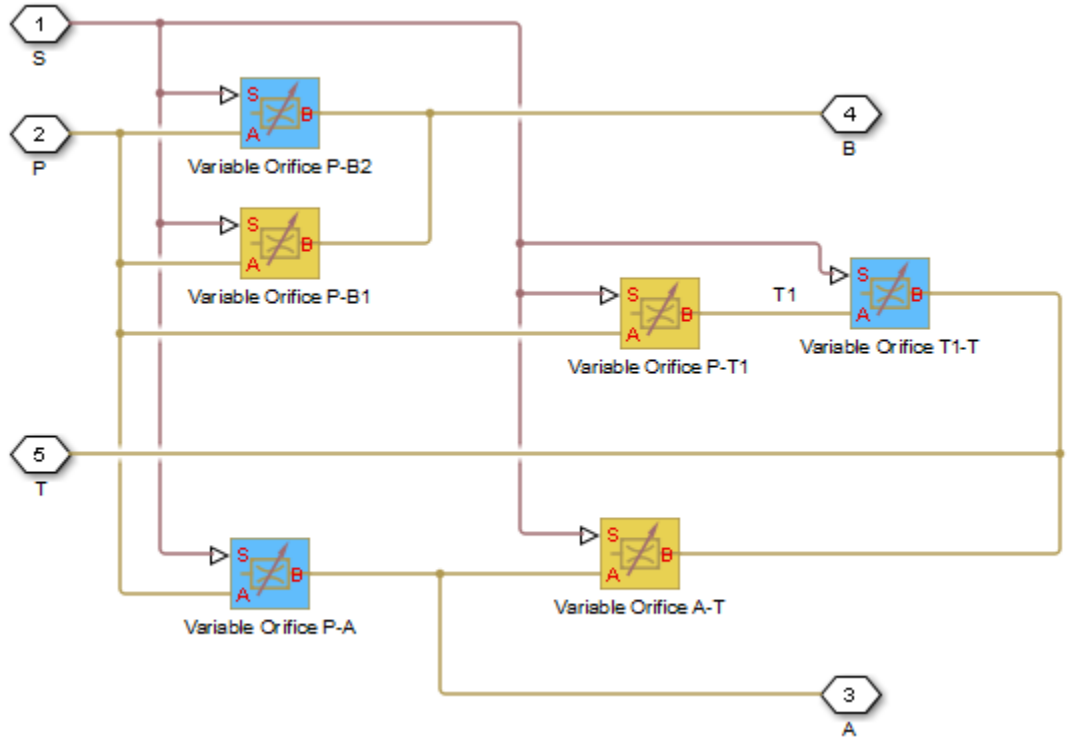
Configuration H of hydraulic continuous 4-way directional valve

Library

Directional Valves

Description

The 4-Way Directional Valve H block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the left valve position. In the right position, port P is connected to port B, while port A is connected to port T. In neutral position, port P is connected to port T to unload the pump. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-B2, P-A, and T1-T) and closes the orifices colored yellow (orifices P-B1, P-T1, and A-T). As a result, the openings of the orifices are computed as follows:

$$h_{PB1} = h_{PB10} - x$$

$$h_{PB2} = h_{PB20} + x$$

$$h_{PT1} = h_{PT10} - x$$

$$h_{T1T} = h_{T1T0} + x$$

$$h_{PA} = h_{PA0} + x$$

$$h_{AT} = h_{AT0} - x$$

where

h_{PB1}	Orifice opening for the `Variable Orifice P-B1` block
h_{PB2}	Orifice opening for the `Variable Orifice P-B2` block
h_{PT1}	Orifice opening for the `Variable Orifice P-T1` block
h_{T1T}	Orifice opening for the `Variable Orifice T1-T` block
h_{PA}	Orifice opening for the `Variable Orifice P-A` block
h_{AT}	Orifice opening for the `Variable Orifice A-T` block
h_{PB10}	Initial opening for the `Variable Orifice P-B1` block
h_{PB20}	Initial opening for the `Variable Orifice P-B2` block
h_{PT10}	Initial opening for the `Variable Orifice P-T1` block
h_{T1T0}	Initial opening for the `Variable Orifice T1-T` block
h_{PA0}	Initial opening for the `Variable Orifice P-A` block
h_{AT0}	Initial opening for the `Variable Orifice A-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-206
- “Valve Opening Offsets Tab” on page 1-210

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector.

All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and B1

Orifice opening of the **P-B1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and B2

Orifice opening of the **P-B2** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and T1

Orifice opening of the **P-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports T1 and T

Orifice opening of the **T1-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports P and A

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports A and T

Orifice opening of the **A-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

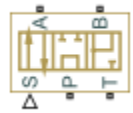
See Also

4-Way Directional Valve C | 4-Way Directional Valve K | 4-Way Directional Valve D | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way Directional Valve E | 4-Way Directional Valve F | 4-Way Directional Valve G | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve K

Configuration K of hydraulic continuous 4-way directional valve

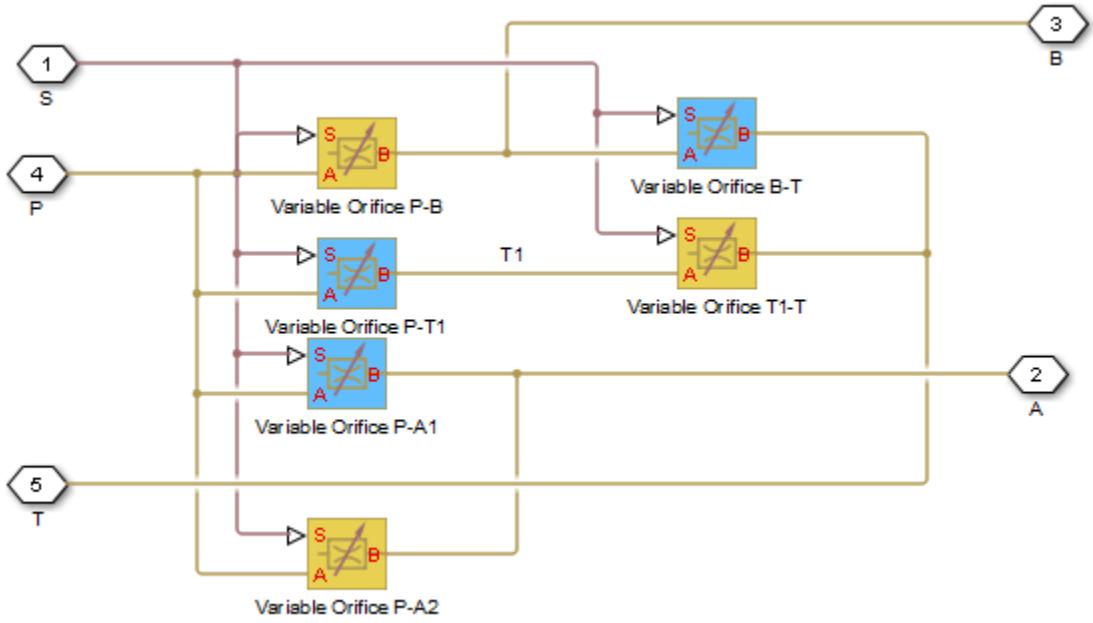


Library

Directional Valves

Description

The 4-Way Directional Valve K block simulates a configuration of hydraulic continuous 4-way directional valve. Ports A and B are connected to port P in the right valve position. In the left position, port P is connected to port B, while port A is connected to port T. In neutral position, port P is connected to port T to unload the pump. The fluid is pumped in the valve through the inlet line P and is distributed between two outside hydraulic lines A and B (usually connected to a double-acting actuator) and the return line T. The block has four hydraulic connections, corresponding to inlet port (P), actuator ports (A and B), and return port (T), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



All Variable Orifice blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A1, P-T1, and B-T) and closes the orifices colored yellow (orifices P-A2, P-B, and T1-T). As a result, the openings of the orifices are computed as follows:

$$h_{PA1} = h_{PA10} + x$$

$$h_{PA2} = h_{PA20} - x$$

$$h_{PT1} = h_{PT10} + x$$

$$h_{T1T} = h_{T1T0} - x$$

$$h_{PB} = h_{PB0} - x$$

$$h_{BT} = h_{BT0} + x$$

where

h_{PA1}	Orifice opening for the `Variable Orifice P-A1` block
h_{PA2}	Orifice opening for the `Variable Orifice P-A2` block
h_{PT1}	Orifice opening for the `Variable Orifice P-T1` block
h_{T1T}	Orifice opening for the `Variable Orifice T1-T` block
h_{PB}	Orifice opening for the `Variable Orifice P-B` block
h_{BT}	Orifice opening for the `Variable Orifice B-T` block
h_{PA10}	Initial opening for the `Variable Orifice P-A1` block
h_{PA20}	Initial opening for the `Variable Orifice P-A2` block
h_{PT10}	Initial opening for the `Variable Orifice P-T1` block
h_{T1T0}	Initial opening for the `Variable Orifice T1-T` block
h_{PB0}	Initial opening for the `Variable Orifice P-B` block
h_{BT0}	Initial opening for the `Variable Orifice B-T` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-214
- “Valve Opening Offsets Tab” on page 1-218

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly

dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e +06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.

- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and A1

Orifice opening of the **P-A1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and A2

Orifice opening of the **P-A2** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and T1

Orifice opening of the **P-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports T1 and T

Orifice opening of the **T1-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports P and B

Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports B and T

Orifice opening of the **B-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

P

Hydraulic conserving port associated with the pressure supply line inlet.

T

Hydraulic conserving port associated with the return line connection.

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

S

Physical signal port to control spool displacement.

See Also

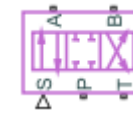
4-Way Directional Valve C | 4-Way Directional Valve H | 4-Way Directional Valve D | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way Directional Valve E | 4-Way Directional Valve F | 4-Way Directional Valve G | 6-Way Directional Valve A

Introduced in R2009b

4-Way Directional Valve (G)

Controlled valve with four ports and four flow paths

Library: Simscape / Fluids / Gas / Valves & Orifices / Directional Control Valves



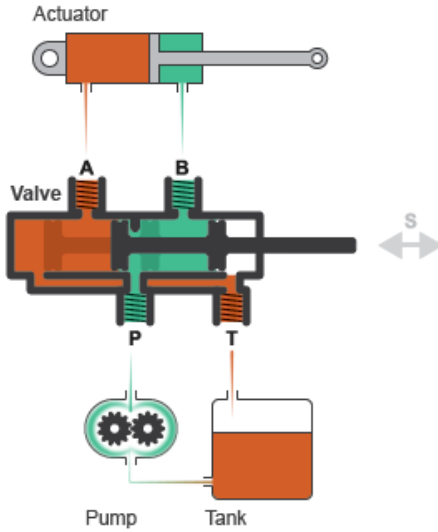
Description

The 4-Way Directional Valve (G) block models a valve with four gas ports (**P**, **A**, **B**, and **T**) and two sets of flow paths to switch between (**P**–**A** and **A**–**T**, **P**–**B** and **B**–**T**). The paths each run through an orifice of variable width, its opening being tied to the position of a control member. Think of the control member as a spool with two lands to cover (by degrees) the different orifices. The distance of a land to its appointed orifices determines if, and to what extent, those are open.

(The distances from the lands to the orifices are computed during simulation from the displacement signal specified at port **S**. They, and all distances related to spool position, are defined as unitless fractions, typically ranging from -1 to +1. The calculations are described in detail under “Orifice Opening Fractions” on page 1-222.)

The gas ports connect to what, in a representative system, are a pump (**P**), a tank (**T**), and a double-sided actuator (**A** and **B**). Opening the **P**–**A** and **B**–**T** flow paths allows the pump to pressurize one side of the actuator and the tank to relieve the other. The actuator shaft translates (to extend in some systems, to retract in others). Opening the alternate (**P**–**B** and **A**–**T**) flow paths flips the pressurized and relieved sides of the actuator so that its shaft can translate in reverse.

The port connections will vary with the system being modeled, but the purpose of the valve—to switch between flow paths and to regulate the flow passing through them—should not.



A Common Use of a 4-Way Directional Valve

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

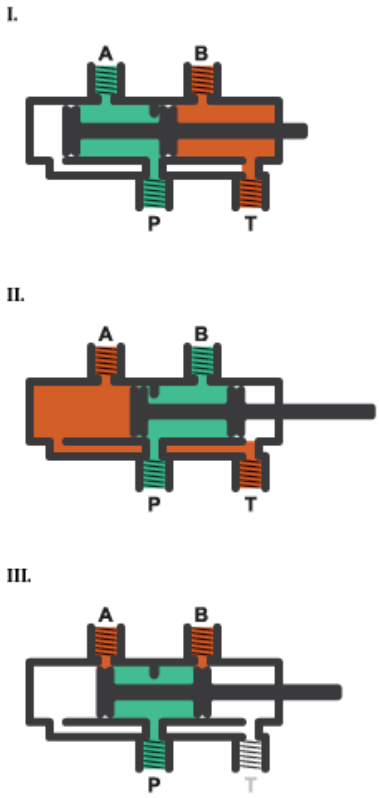
Valve Positions

The valve is continuously variable. It shifts smoothly between positions, of which it has three: one normal and two working.

The normal position is that to which the valve reverts when it is no longer being operated. The instantaneous displacement of the spool (given at port **S**) is then zero. Unless the lands of the spool are installed at an offset to their orifices, the valve will be fully closed.

The other two, the working positions, are those to which the valve moves when the spool is maximally displaced from its normal position. That displacement is positive in one case and negative in the other.

If it is positive, the **P**–**A** and **B**–**T** orifices are fully open and the **P**–**B** and **A**–**T** orifices are fully closed (position **I** in the figure). If it is negative, the **P**–**A** and **B**–**T** orifices are fully closed and the **P**–**B** and **A**–**T** orifices are fully open (position **II**).



What spool displacement puts the valve in a working position depends on the offsets of the lands on the spool. These are generally applied before operation, in a real valve, and before simulation, in a valve model. They are specified in the block as constants (fixed from the start of simulation) in the **Valve Opening Fraction Offsets** tab.

Between valve positions, the opening of an orifice depends on where, relative to its rim, its land of the spool happens to be. This distance is the orifice opening, and it is normalized here so that its value is a fraction of its maximum (the distance at which the

orifice is fully open). The normalized variable is referred to here as the *orifice opening fraction*.

The orifice opening fractions range from `-1` in working position **I** to `+1` in working position **II** (using the labels shown in the figure).

The opening fractions are calculated from the lengths already alluded to: the variable displacement of the control member (applied during operation) and the fixed offsets of its lands (applied during installation). These lengths are themselves defined as unitless fractions of the maximum land–orifice distance. (The offsets are referred to here as the *opening fraction offsets*.)

The opening fraction of the **P**–**A** orifice is:

$$h_{\mathbf{PA}} = H_{\mathbf{PA}} + x.$$

Likewise for the **B**–**T** orifice:

$$h_{\mathbf{BT}} = H_{\mathbf{BT}} + x.$$

That of the **A**–**T** orifice is:

$$h_{\mathbf{AT}} = H_{\mathbf{AT}} - x.$$

Lastly for the **P**–**B** orifice:

$$h_{\mathbf{PB}} = H_{\mathbf{PB}} - x.$$

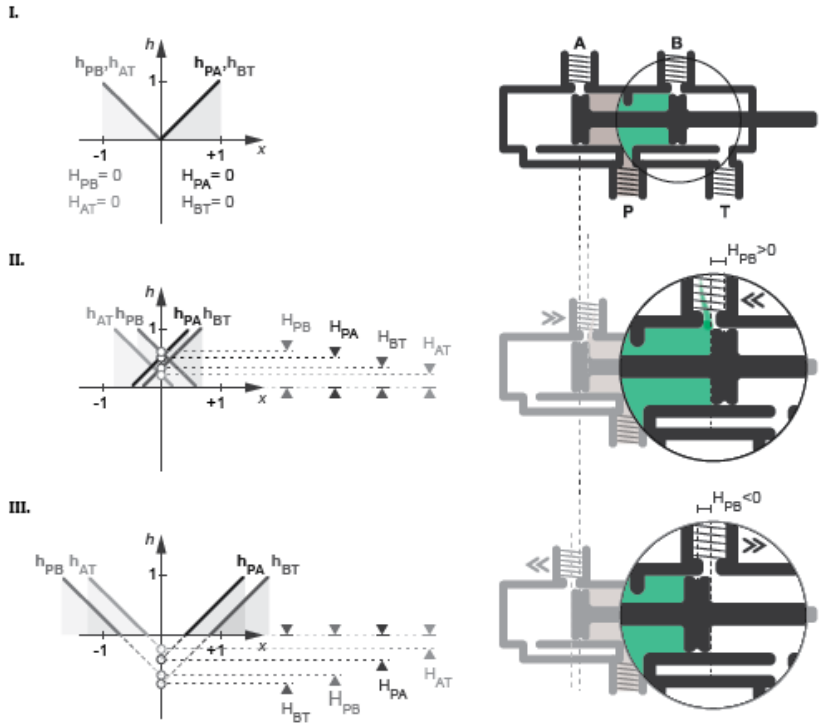
In the equations:

- h is the opening fraction of the orifice denoted by the subscript. If the calculation should return a value outside of the range **0**–**1**, the nearest limit is used. (The orifice opening fractions are said to *saturate* at `0` and `1`.)
- H is the opening fraction offset for the orifice denoted by the subscript. The offsets are each specified as a block parameter (in the **Valve Opening Fraction Offsets** tab). To allow for unusual valve configurations, no limit is imposed on their values, though generally these will fall between `-1` and `+1`.
- x is the normalized instantaneous displacement of the spool, specified as a physical signal at port **S**. To compensate for equally extreme opening fraction offsets, no limit is imposed on its value (though generally it too will fall in the range of `-1` to `+1`.)

The valve is by default configured so that it is fully closed when the spool displacement is zero. Such a valve is often described as being *zero-lapped*.

It is possible, by offsetting the lands of the spool, to model a valve that is underlapped (partially open in the normal valve position) or overlapped (fully closed not only *in* but also slightly *beyond* the normal position). The figure shows, for each case, how the orifice opening fractions vary with the instantaneous spool displacement:

- Case **I**: A zero-lapped valve. The opening fraction offsets are all zero. When the valve is in the normal position, the lands of the spool completely cover all four orifices.
- Case **II**: An underlapped valve. The **P**–**A** and **B**–**T** opening fraction offsets are positive and the **P**–**B** and the **A**–**T** opening fraction offsets are negative. When the valve is in the normal position, the lands of the spool cover all four orifices, but neither fully.
- Case **III**: An overlapped valve. The **P**–**A** and **B**–**T** opening fraction offsets are negative and the **P**–**B** and the **A**–**T** opening fraction offsets are positive. The control member completely covers all orifices not only in the normal position but over a small region (of spool displacements) around it.



Opening Characteristics

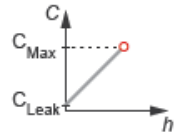
It is common, when picking a valve for throttling or control applications, to match the flow characteristic of the valve to the system it is to regulate.

The flow characteristic relates the opening of the valve to the input that produces it, often spool travel. Here, the opening is expressed as a sonic conductance, flow coefficient, or restriction area (the choice between them being given by the **Valve parameterization** setting). The control input is the orifice opening fraction (a function of the spool displacement specified at port **S**).

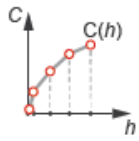
The flow characteristic is normally given at steady state, with the inlet at a constant, carefully controlled pressure. This (inherent) flow characteristic depends only on the valve and it can be linear or nonlinear, the most common examples of the latter being the quick-opening and equal-percentage types. To capture such flow characteristics, the block

provides a choice of opening parameterization (specified in the block parameter of the same name):

- `Linear` — The sonic conductance (C) is a linear function of the orifice opening fraction (h). In the default valve parameterization of `Sonic conductance`, the end points of the line are obtained at opening fractions of `0` and `1` from the **Sonic conductance and leakage flow** and **Sonic conductance at maximum flow** block parameters.



- `Tabulated data` — The sonic conductance is a general function (linear or nonlinear) of the orifice opening fraction. The function is specified in tabulated form, with the columns of the table deriving, in the default valve parameterization, from the **Opening fraction vector** and **Sonic conductance vector** block parameters.



(If the **Valve parameterization** setting is other than `Sonic conductance`, the sonic conductance data is obtained by conversion from the chosen measure of valve opening (such as restriction area or flow coefficient). The opening data applies to all orifices equally.)

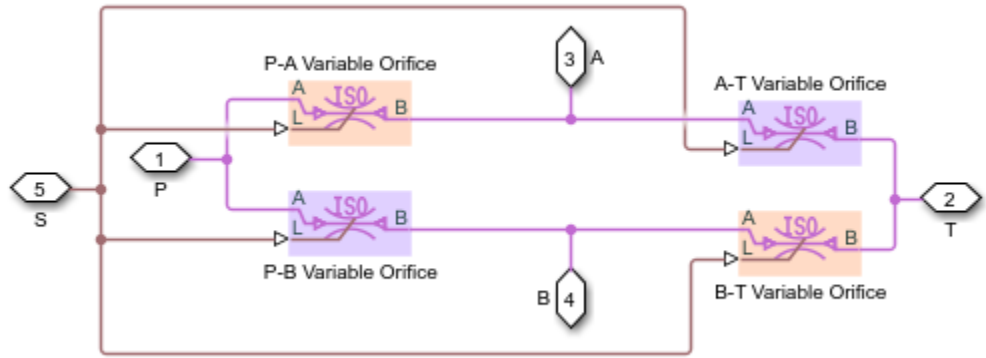
For controlled systems, it is important that the valve, once it is installed, be approximately linear in its flow characteristic. This (installed) characteristic depends on the remainder of the system—it is not generally the same as the inherent characteristic captured in the block. A pump, for example, may have a nonlinear characteristic that only a nonlinear valve, usually of the equal-percentage type, can adequately compensate for. It is cases of this sort that the `Tabulated data` option primarily targets.

Leakage Flow

The main purpose of leakage flow is to ensure that no section of a fluid network ever becomes isolated from the rest. Isolated fluid sections can reduce the numerical robustness of the model, slowing down the rate of simulation and, in some cases, causing it to fail altogether. While leakage flow is generally present in real valves, its exact value here is less important than its being a small number greater than zero. The leakage flow area is given in the block parameter of the same name.

Composite Structure

This block is a composite component comprising two instances of the Variable Orifice ISO 6358 (G) block connected to ports **P**, **A**, **T**, and **S** as shown below. Refer to that block for more detail on the valve parameterizations and block calculations (for example, those used to determine the mass flow rate through the ports).



Ports

Input

S — Valve control signal, `unitless`

physical signal

Instantaneous displacement of the control member against its normal (unactuated) position, specified as a physical signal. The displacement is normalized against the

maximum position of the control member (that required to open the orifice fully). See the block description for more information. The signal is unitless and its instantaneous value typically (though not always) in the range of -1–+1.

Conserving

A — Valve opening
gas

Opening through which flow can enter or exit the valve.

B — Valve opening
gas

Opening through which flow can enter or exit the valve.

P — Valve opening
gas

Opening through which flow can enter or exit the valve.

T — Valve opening
gas

Opening through which flow can enter or exit the valve.

Parameters

Basic Parameters

`Valve parameterization` — Method by which to characterize the opening of the valve
`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the `Sonic conductance` parameterization; if a different option is selected, the data specified is converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the calculations of the Variable Orifice ISO 6358 (G) block for detail on the conversions.

`Opening parameterization` — Method by which to calculate the opening characteristics of the valve
`Linear` (default) | `Tabulated data`

Method by which to calculate the opening area of the valve. The default setting treats the opening area as a linear function of the orifice opening fraction. The alternative setting allows for a general, nonlinear relationship to be specified (in tabulated form).

`Cross-sectional area at ports A, B, P, and T` — Area normal to the flow path at the valve ports
`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be of the same size. The flow area specified here should (ideally) match those of the inlets of adjoining components.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes
`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance
`293.15 K` (default) | scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance
`1.185` (default) | positive scalar in units of mass/volume

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

Model Parameterization

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions

`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value usually reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions

`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum
`0.3` (default) | positive and unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime
`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units
`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of the leakage value is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of L/min. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of L/min. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of the leakage value is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause

problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Restriction area at maximum flow` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Sum of the gauge pressures at the inlet and pilot port at which the valve is fully open. This value marks the end of the pressure range of the valve (over which the same progressively opens to allow for increased flow).

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Restriction area at leakage flow` — Opening area in the maximally closed position due to sealing imperfections

`1e-10` (default)

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Opening fraction vector` — Orifice opening fractions at which to specify valve opening data

`0 : 0.2 : 1` (default) | unitless vector with elements in the `0`–`1` range

Orifice opening fractions at which to specify the chosen measure of valve opening—sonic conductance, flow coefficient (in SI or USCS forms), or opening area.

This vector must be equal in size to that (or *those*, in the `Sonic conductance` parameterization) containing the valve opening data. The vector elements must be positive and increase monotonically in value from left to right.

Dependencies

This parameter is active and exposed in the block dialog box when the **Opening parameterization** setting is `Tabulated data`.

`Sonic conductance vector` — Vector of sonic conductances at given orifice opening fractions

`[1e-05, .32, .64, .96, 1.28, 1.6] l/s/bar` (default) | vector with units of volume/time/pressure

Sonic conductances at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the orifice opening fraction, sonic conductance, and critical pressure ratio. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of sonic conductance and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Critical pressure ratio vector` — Vector of critical pressure ratios at given orifice opening fractions

`0.3 * ones(1, 6)` (default) | unitless vector

Critical pressure rations at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the orifice opening fraction, sonic conductance, and critical pressure ratio. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of critical pressure ratios and orifice opening fractions—must be of the same size.

The values specified here must each be greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Cv coefficient (USCS) vector` — Vector of flow coefficients, in USCS units, at given orifice opening fractions

`[1e-06, .08, .16, .24, .32, .4]` (default) | vector in units of ft^3/min

Flow coefficients, expressed in US customary units of ft^3/min, at the breakpoints given in the **Opening fraction vector**. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of flow coefficients and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)` and the **Opening parameterization** setting is `Tabulated data`.

`Kv coefficient (SI) vector` — Vector of flow coefficients, in SI units, at given orifice opening fractions

`[1e-06, .06, .12, .18, .24, .3]` (default) | vector in units of L/min

Flow coefficients, expressed in SI units of `L/min`, at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of flow coefficients and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)` and the **Opening parameterization** setting is `Tabulated data`.

`Restriction area vector` — Vector of opening areas at given orifice opening fractions

`[1e-10, 2e-06, 4e-06, 6e-06, 8e-06, 1e-05] m^2` (default) | vector in units of area

Opening areas at the breakpoints given in the **Opening fraction vector** parameter. This data forms the basis for a tabulated function relating the two variables. Linear interpolation is used within the tabulated data range; nearest-neighbor extrapolation is used outside of it. The two vectors—of opening areas and orifice opening fractions—must be of the same size.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area` and the **Opening parameterization** setting is `Tabulated data`.

`Smoothing factor` — Amount of smoothing to apply to the valve opening function

`0` (default) | positive unitless scalar

Amount of smoothing to apply to the opening function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become. See the Variable Orifice ISO 6358 (G) block for the impact of the smoothing on the block calculations.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings). These can slow down the rate of simulation.

Valve Opening Fraction Offsets

`Between Ports P and A` — Opening fraction offset for the P–A orifice

`0` (default) | unitless scalar

Opening fraction of the **P**–**A** orifice when the spool displacement is zero. The valve is then in the normal position. The opening fraction measures the distance of a land of the spool to its appointed orifice (here **P**–**A**), normalized by the maximum such distance. It is unitless and (generally) between `0` and `1`.

`Between Ports B and T` — Opening fraction offset for the B–T orifice

`0` (default) | unitless scalar

Opening fraction of the **B**–**T** orifice when the spool displacement is zero. The valve is then in the normal position. The opening fraction measures the distance of a land of the spool to its appointed orifice (here **B**–**T**), normalized by the maximum such distance. It is unitless and (generally) between `0` and `1`.

`Between Ports P and B` — Opening fraction offset for the P–B orifice
`0` (default) | unitless scalar

Opening fraction of the **P**–**B** orifice when the spool displacement is zero. The valve is then in the normal position. The opening fraction measures the distance of a land of the spool to its appointed orifice (here **P**–**B**), normalized by the maximum such distance. It is unitless and (generally) between `0` and `1`.

`Between Ports A and T` — Opening fraction offset for the A–T orifice
`0` (default) | unitless scalar

Opening fraction of the **A**–**T** orifice when the spool displacement is zero. The valve is then in the normal position. The opening fraction measures the distance of a land of the spool to its appointed orifice (here **A**–**T**), normalized by the maximum such distance. It is unitless and (generally) between `0` and `1`.

See Also

2-Way Directional Valve (G) | 3-Way Directional Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

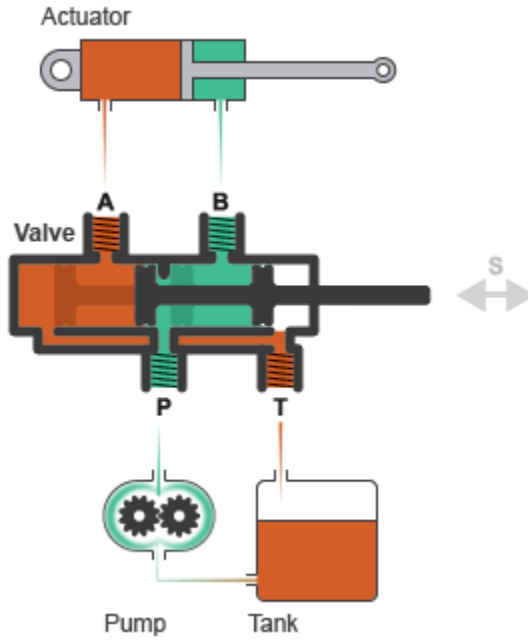
4-Way Directional Valve (TL)

Valve for routing flow at the junction of four lines

Library: Simscape / Fluids / Thermal Liquid / Valves & Orifices / Directional Control Valves

Description

The 4-Way Directional Valve block models the flow through a directional control valve with four ports (**P**, **T**, **A**, and **B**) and four flow paths (**P**-**A**, **P**-**B**, **A**-**T**, and **B**-**T**). The ports connect to what in a typical system are a pump (**P**), a tank (**T**), and a double-acting actuator (**A** and **B**). The paths each contain a variable orifice, which scales in proportion to the displacement of a control member—often a ball, spool, or diaphragm, associated with the signal at port **S**. This valve serves as a switch by which to partition the flow between the arms of a four-pronged junction.



Typical Valve Setup

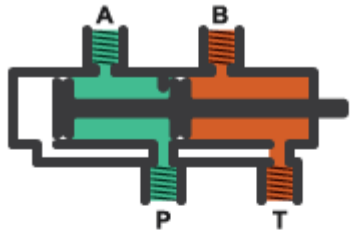
Valve Positions

The valve is continuously variable. It shifts smoothly between positions, of which it has three. One—the *normal* position—is that to which the valve reverts when its control signal falls to zero. Unless opening offsets have been specified, the valve orifices are always fully closed in this position. Two—the *working* positions—are those to which the valve moves when (the absolute value) of its control signal rises to a maximum. Two orifices are generally fully closed and another two fully open when the valve is in a working position. Note that whether an orifice is in fact open and how open it is both depend on the opening offsets of the valve.

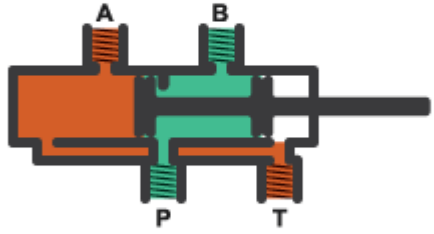
The working positions are shown in the figure in the default case of a valve without opening offsets. One, labeled **I**, corresponds to the **P**-**A** and **B**-**T** orifices being maximally open and the **P**-**B** and **A**-**T** orifices maximally closed. The other, labeled **II**, corresponds to the reverse arrangement, with the **P**-**B** and **A**-**T** orifices being maximally open and the **P**-**A** and **B**-**T** orifices maximally closed. At points between the normal and working positions,

one orifice is partially open while the other is fully closed. Note that no connection exists between ports **A** and **B** nor between **P** and **T** and that no flow can develop across them.

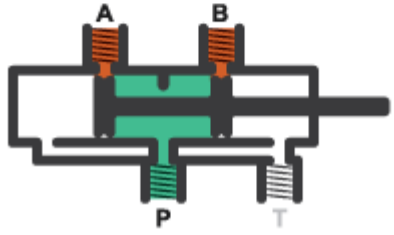
I.



II.



III.



Valve Opening

Which position the valve is in depends on the control member coordinates relative to the **P**-**A**, **P**-**B**, **A**-**T**, and **B**-**T** orifices—lengths referred to here as the *orifice openings*. These are calculated during simulation from their opening offsets, each specified as a block

parameter in the block dialog box, and from the control member displacement, given by the physical signal at port **S**. For the **P**-**A** and **B**-**T** orifices:

$$h_{PA} = h_{PA0} + x \text{ and } h_{BT} = h_{BT0} + x,$$

where:

- h_{PA} and h_{BT} are the **P**-**A** and **B**-**T** orifice openings.
- h_{PA0} and h_{BT0} are the **P**-**A** and **B**-**T** opening offsets.
- x is the control member displacement. Note that a control member displacement of zero corresponds to a valve in its normal state.

For the **P**-**B** and **A**-**T** orifices:

$$h_{PB} = h_{PB0} + x \text{ and } h_{AT} = h_{AT0} + x,$$

where:

- h_{PB} and h_{AT} are the **P**-**B** and **A**-**T** orifice openings.
- h_{PB0} and h_{AT0} are the **P**-**B** and **A**-**T** opening offsets.

An orifice cracks opens when its calculated opening (variable h) rises above zero. It then continues to widen with a rising opening value. In the cases of the **P**-**B** and **A**-**T** orifices, it occurs as the control member is displaced in the negative direction.

The orifices are each fully open when the opening value is at a specified maximum. In the linear valve parameterization, this maximum is obtained from the **Maximum valve opening** block parameter. You can set the **Area characteristics** parameter to `Different for each flow path` to specify the **Maximum valve opening** parameter separately for each orifice. In tabulated valve parameterizations, the maximum opening is obtained from the last breakpoint specified in the tabulated data.

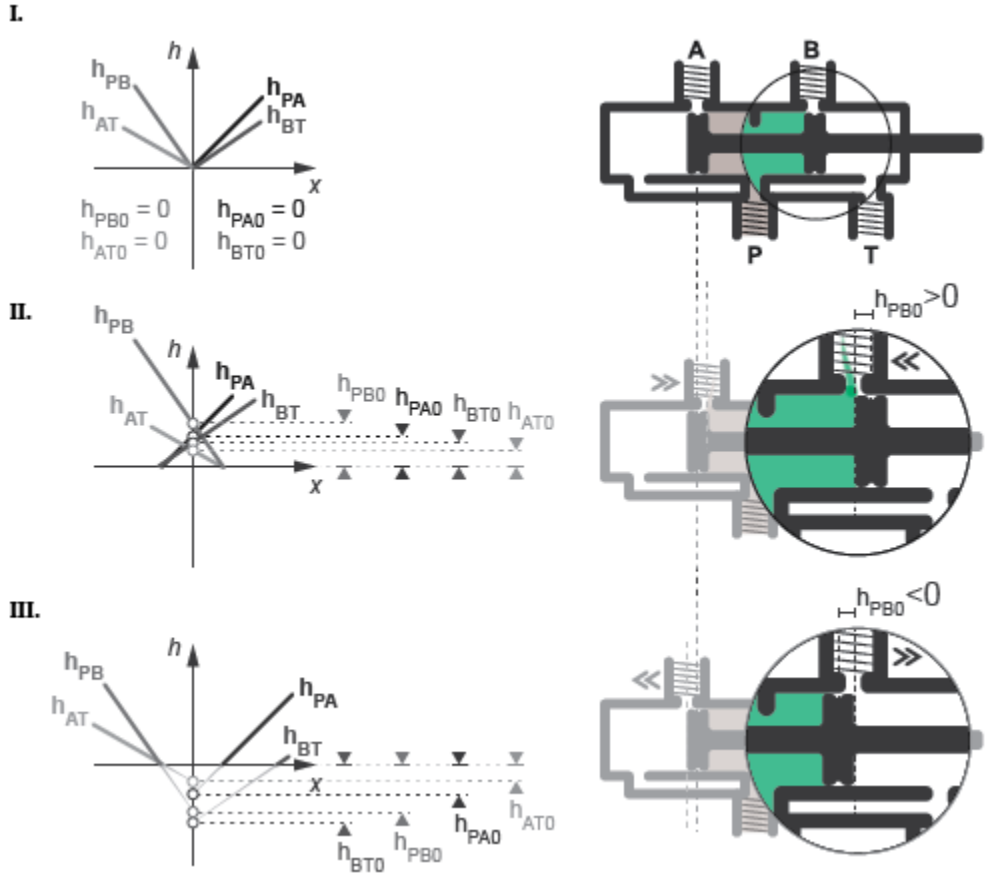
Opening Offsets

The opening offsets are by default zero, a configuration that leaves the orifices each closed in the normal valve position. A valve so configured is said to be *zero-lapped*, a reference to the fact that the control member is exactly sized to prevent flow through any orifice when idle, or unactuated ($x = 0$). For the purposes of this block, the opening offsets can be conceived as the natural distances between the lands of the (unactuated) control member and the orifices that they are to cover.

Other valve configurations are possible. A valve can be *underlapped*, for example, or *overlapped*—terms indicative of the mismatch between the dimensions of the control member and those of the valve orifices. The valve is underlapped if its orifices are each partially open in the normal position. Such a valve allows a weak flow to develop across all paths simultaneously. The valve is overlapped if its orifices are each closed not only in the normal position but over a small range of control member displacements around it.

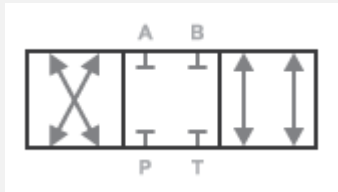
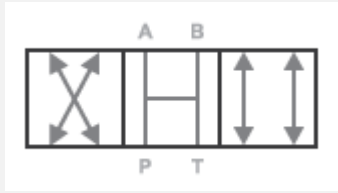
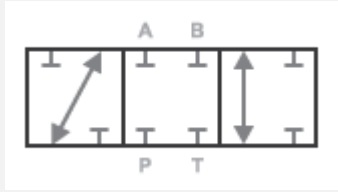
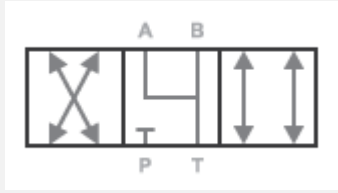
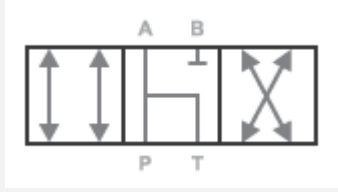
The figure shows a representative valve in three configurations:

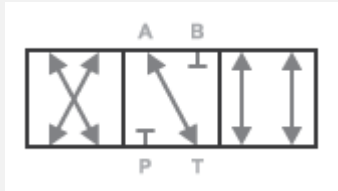
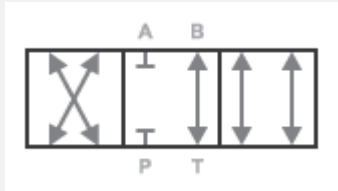
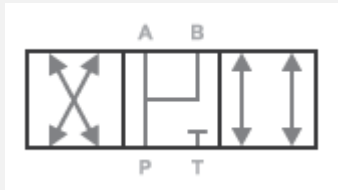
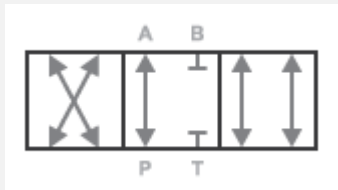
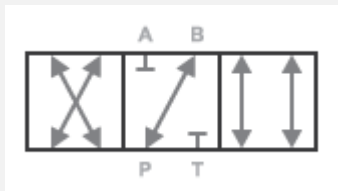
- Case **I**: A zero-lapped valve. The opening offsets are both zero. When the valve is in the normal position, the control member completely covers both orifices. The zero-lapped valve is completely closed when the control member displacement is exactly zero.
- Case **II**: An underlapped valve. The opening offsets are both positive. When the valve is in the normal position, the control member covers both orifices but neither fully. The underlapped valve is always at least partially open.
- Case **III**: An overlapped valve. The opening offsets are both negative. The control member completely covers both orifices not only in the normal position but over a small region around it. The overlapped valve is fully closed until the control member crosses the opening offset of either orifice.

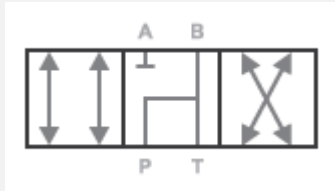


The table summarizes the configurations that the valve can take on and the opening offsets that characterize them. Use the block parameters in the **Valve opening offsets** tab to change the offsets if needed.

Basic 4-Way Directional Valve Configurations

Configuration	Valve opening offsets
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0
	• **Between ports P and A** > 0 • **Between ports P and B** > 0 • **Between ports A and T** > 0 • **Between ports B and T** > 0
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** < `-Maximum valve opening` • **Between ports B and T** < `-Maximum valve opening`
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** > 0 • **Between ports B and T** > 0
	• **Between ports P and A** > 0 • **Between ports P and B** < 0 • **Between ports A and T** > 0 • **Between ports B and T** < 0

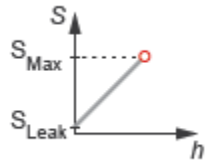
Configuration	Valve opening offsets
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** > 0 • **Between ports B and T** < 0
	• **Between ports P and A** < 0 • **Between ports P and B** < 0 • **Between ports A and T** < 0 • **Between ports B and T** > 0
	• **Between ports P and A** > 0 • **Between ports P and B** > 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0
	• **Between ports P and A** > 0 • **Between ports P and B** < 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0
	• **Between ports P and A** < 0 • **Between ports P and B** > 0 • **Between ports A and T** < 0 • **Between ports B and T** < 0

Configuration	Valve opening offsets
	• **Between ports P and A** < 0 • **Between ports P and B** > 0 • **Between ports A and T** < 0 • **Between ports B and T** > 0

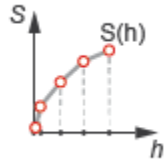
Opening Characteristics

The orifice openings serve during simulation to calculate the mass flow rates through the orifices. The calculation can be a direct mapping from opening to flow rate or an indirect conversion, first from opening to orifice area and then from orifice area to mass flow rate. The calculation, and the data required for it, depend on the setting of the **Valve parameterization** block parameter:

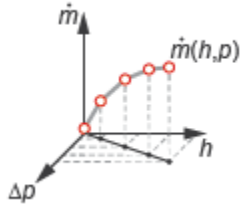
- `Linear area-opening relationship` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve. The opening area is assumed to vary linearly with the control member position. The slope of the linear expression is determined from the **Maximum valve opening** and **Maximum opening area** block parameters.



- `Tabulated data - Area vs. opening` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve. The opening area can vary nonlinearly with the control member position. The relationship between the two is given by the tabulated data in the **Valve opening vector** and **Opening area vector** block parameters.



- `Tabulated data - Mass flow rate vs. opening and pressure drop` — Calculate the mass flow rate directly from the control member position and the pressure drop across the valve. The relationship between the three variables can be nonlinear and it is given by the tabulated data in the **Valve opening vector**, **Pressure drop vector**, and **Mass flow rate table** block parameters.



Leakage Flow

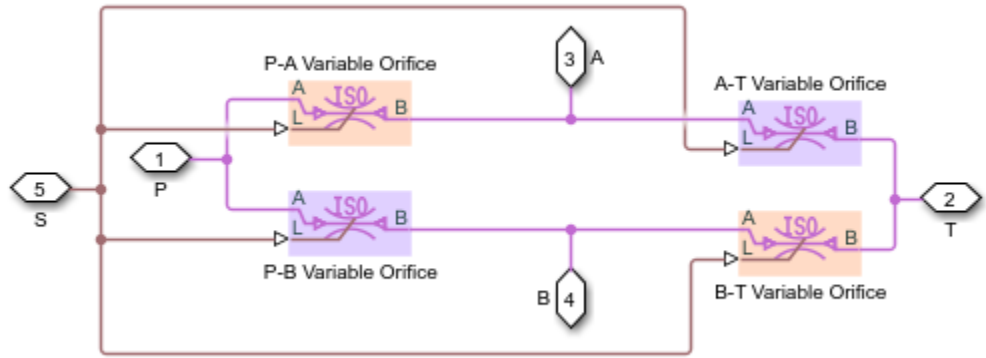
The primary purpose of the leakage flow rate of a closed valve is to ensure that at no time a portion of the thermal liquid network becomes isolated from the remainder of the model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage flow is generally present in real valves but in a model its exact value is less important than its being a small number greater than zero. The leakage flow rate is determined from the **Leakage area** block parameter.

Pressure Loss and Recovery

The pressure drop in the valve is calculated from an empirical parameter known as the discharge coefficient (obtained from the **Discharge coefficient** block parameter). The calculation captures the effect of flow regime, with the pressure drop being proportional to the mass flow rate when the flow is laminar and to the square of the same when the flow is turbulent. Also captured is the pressure recovery than in real valves occurs between the vena contracta (the narrowest aperture of the valve) and the outlet, which generally lies a small distance away.

Composite Component Structure

This block is a composite component comprising four Variable Area Orifice (TL) blocks connected as shown in the figure. A single control signal actuates the four blocks simultaneously. The **Orifice orientation** block parameters are set so that a positive signal acts to open the **P**-**A** and **B**-**T** orifices while closing the **P**-**B** and **A**-**T** orifices. The specified opening offsets are each applied to the block representing the intended orifice. Refer to the Variable Area Orifice (TL) block for detail on the opening area calculations.



Ports

Input

S — Control member displacement, `unitless`

physical signal

Instantaneous displacement of the valve control member.

Conserving

P — Flow passage

thermal liquid

Opening through which the flow can enter or exit the valve. This port is typically connected to a fluid supply line.

T — Flow passage
thermal liquid

Opening through which the flow can enter or exit the valve. This port is typically connected to a fluid return line.

A — Flow passage
thermal liquid

Opening through which the flow can enter or exit the valve. This port is typically connected to an actuation line.

B — Flow passage
thermal liquid

Opening through which the flow can enter or exit the valve. This port is typically connected to an actuation line.

Parameters

Basic Parameters

`Area characteristics` — Choice of identical or different orifice opening characteristics
`Identical for all flow paths` (default) | `Different for each flow path`

Assumption to make in the opening characteristics of the valve orifices. The default setting attributes the same opening characteristics to each orifice. The maximum opening area and the orifice opening at which it occurs are then the same for all orifices. Use the alternate setting to specify these parameters separately for each orifice.

`Valve parameterization` — Method by which to model the opening characteristics of the valve
`Linear area-opening relationship` (default) | `Tabulated data - Area vs. opening` | `Tabulated data - Mass flow rate vs. opening and pressure drop`

Method by which to model the opening characteristics of the valve. The default setting prescribes a linear relationship between the valve opening area and the valve opening. The alternative settings allow for a general, nonlinear relationship to be specified in tabulated form, in one case between the opening area and the control member position

(the valve opening), in the other case between the mass flow rate and both the valve opening and the pressure drop between the ports.

Leakage area — Opening area of the valve in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

Cross-sectional area at ports A, B, P, and T — Flow area at the thermal liquid ports

`0.01 m` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the valve connects.

Characteristic longitudinal length — Measure of the length of the flow path through the valve

`0.1` (default) | positive scalar in units of length

Average distance traversed by the fluid as it travels from inlet to outlet. This distance is used in the calculation of the internal thermal conduction that occurs between the two ports (as part of the smoothed upwind energy scheme employed in the thermal liquid domain).

Discharge coefficient — Empirical factor defined as the ratio of actual to ideal mass flow rates

`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes
`12` (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

Model Parameterization

`Maximum valve opening` — Control member position at which the opening area is at a maximum
`5e-3 m` (default) | positive scalar in units of length

Value of the orifice opening at which an orifice is considered to be fully open. The opening area of the orifice is then at a maximum. The maximum opening is used to calculate the slope of the linear expression relating the orifice opening area to the orifice opening.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Maximum opening area` — Opening area of the valve in the fully open position
`1e-4 m^2` (default) | positive scalar in units of area

Opening area of an orifice when its orifice opening has the value specified in the **Maximum valve opening** block parameter. The maximum opening area parameter is used to calculate the slope of the linear expression relating the orifice opening area to the orifice opening.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Smoothing factor` — Measure of the amount of smoothing to apply to the opening area function
`0.01` (default) | positive unitless scalar

Measure of the amount of smoothing to apply to the opening area function. This parameter determines the widths of the regions to be smoothed, one being at the fully open position, the other at the fully closed position. The smoothing superposes on the linear opening area function two nonlinear segments, one for each region of smoothing.

The greater the value specified, the greater the smoothing and the broader the nonlinear segments.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Linear area-opening relationship`.

`Valve opening vector` — Vector of control member positions at which to tabulate the valve opening areas

`[-0.002, 0, 0.002, 0.005, 0.015] m` (default) | vector of positive numbers in units of length

Vector of control member positions at which to specify—dependent on the valve parameterization—the opening area of the valve or its mass flow rate. The vector elements must increase monotonically from left to right. This vector must be equal in size to that specified in the **Opening area vector** block parameter or to the number of rows in the **Mass flow rate table** block parameter.

This data serves to construct a one-way lookup table by which to determine, from the control member position, the opening area of the valve or a two-way lookup table by which to determine, from the control member position and pressure drop, the mass flow rate of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Opening area vector` — Vector of opening areas at the specified valve opening breakpoints

`[1e-09, 2.0352e-07, 4.0736e-05, 0.00011438, 0.00034356] m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Valve opening vector** block parameter. The vector elements must increase monotonically from left to right (with increasing control member position). This vector must be equal in size to the number of valve opening breakpoints.

This data serves to construct a one-way lookup table by which to determine from the control member position the opening area of the valve. Data is handled with linear

interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Pressure drop vector` — Vector of pressure drops at which to tabulate the mass flow rates

`[-0.1, 0.3, 0.5, 0.7] MPa` (default) | vector in units of pressure

Vector of pressure differentials from port **A** to port **B** at which to specify the mass flow rate of the valve. The vector elements must increase monotonically from left to right. This vector must be equal in size to the number of columns in the **Mass flow rate table** block parameter.

This data serves to construct a two-way lookup table by which to determine, from the control member position and pressure drop, the opening area of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Mass flow rate table` — Matrix of mass flow rates at the specified valve opening and pressure drop breakpoints

`[-1e-05, 1.7e-05, 2e-05, 2.6e-05; -.002, .0035, .0045, .0053; -.4, .7, .9, 1.06; -1.13, 1.96, 2.5, 3; -3.45, 6, 7.7, 9.13] kg/s` (default) | matrix in units of mass/time

Matrix of mass flow rates corresponding to the breakpoints defined in the **Valve opening vector** and **Pressure drop vector** block parameters. The control member position increases from row to row from top to bottom. The pressure drop increases from column to column from left to right. The mass flow rate must increase monotonically in the same directions (with increasing control member position and increasing pressure drop).

This data serves to construct a two-way lookup table by which to determine, from the control member position and pressure drop, the opening area of the valve. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range). Ensure that the number of rows is equal to the

size of the **Opening area vector** block parameter and that the number of columns is equal to the size of the **Pressure drop vector** block parameter.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow temperature` — Inlet temperature at which to specify the tabulated data

`293.15 K` (default) | positive scalar in units of temperature

Nominal inlet temperature, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the temperature measured during simulation.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow pressure` — Inlet pressure at which to specify the tabulated data

`0.101325 MPa` (default) | positive scalar in units of pressure

Nominal inlet pressure, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the pressure measured during simulation.

Dependencies

This parameter is active when the **Valve parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

Valve Opening Offsets

`Between P and A` — Control member offset from the centered position relative to the P-A orifice

`0 m` (default) | positive scalar in units of length

Distance by which to offset the valve control member from its centered (zero) position relative to the **P**-**A** orifice. This parameter determines the control member *displacement*, specified via port **S**, at which the **P**-**A** orifice is fully closed. Note that the control member *position* at which the valve is fully closed is always at zero. Set all opening offsets to

positive values to model an underlapped valve or to negative values to model an overlapped valve.

`Between B and T` — Control member offset from the centered position relative to the P-A orifice
`0 m` (default) | positive scalar in units of length

Distance by which to offset the valve control member from its centered (zero) position relative to the **B**-**T** orifice. This parameter determines the control member *displacement*, specified via port **S**, at which the **B**-**T** orifice is fully closed. Note that the control member *position* at which the valve is fully closed is always at zero. Set all opening offsets to positive values to model an underlapped valve or to negative values to model an overlapped valve.

`Between P and B` — Control member offset from the centered position relative to the P-A orifice
`0 m` (default) | positive scalar in units of length

Distance by which to offset the valve control member from its centered (zero) position relative to the **P**-**B** orifice. This parameter determines the control member *displacement*, specified via port **S**, at which the **P**-**B** orifice is fully closed. Note that the control member *position* at which the valve is fully closed is always at zero. Set all opening offsets to positive values to model an underlapped valve or to negative values to model an overlapped valve.

`Between A and T` — Control member offset from the centered position relative to the A-T orifice
`0 m` (default) | positive scalar in units of length

Distance by which to offset the valve control member from its centered (zero) position relative to the **A**-**T** orifice. This parameter determines the control member *displacement*, specified via port **S**, at which the **A**-**T** orifice is fully closed. Note that the control member *position* at which the valve is fully closed is always at zero. Set all opening offsets to positive values to model an underlapped valve or to negative values to model an overlapped valve.

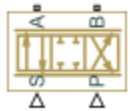
See Also

2-Way Directional Valve (TL) | 3-Way Directional Valve (TL) | Check Valve (TL) | Variable Area Orifice (TL)

Introduced in R2016a

4-Way Ideal Valve

Hydraulic 4-way critically-centered valve



Library

Directional Valves

Description

The 4-Way Ideal Valve block represents a 4-way critically-centered valve, where initial openings of all four variable orifices are equal to zero. This significantly simplifies the model of a 4-way directional valve and makes it especially suitable for real-time and HIL (hardware-in-the-loop) simulation, where such assumption is applicable.

The flow rate is computed with the equation:

$$q = xC_D b\sqrt{\frac{1}{\rho}\left(p_S - abs\left(p_A - p_B\right)\right)}sign\left(p_A - p_B\right)$$

where

q	Flow rate
x	Valve displacement, $-x_{max} <= x <= x_{max}$
b	Orifice width, $b = A_{max} / x_{max}$
A_{max}	Maximum orifice area
x_{max}	Valve maximum opening
C_D	Flow discharge coefficient

ρ	Fluid density
p_S	Pressure supply
p_A,p_B	Pressures at the load ports A and B, respectively

Connections A and B are conserving hydraulic ports associated with the valve load ports. Connections P and S are the physical signal input ports that provide supply pressure and valve displacement values, respectively.

Basic Assumptions and Limitations

- The valve is of a critically-centered type, that is, all initial openings are equal to zero.
- The return pressure is assumed to be very low and can be treated as a zero pressure.
- All the orifices are assumed to have the same shape and size, that is, the valve is symmetrical.

Parameters

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `0.005` m.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Global Parameters

Parameter determined by the type of working fluid:

- **Fluid density**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the actuator connection port.

B

Hydraulic conserving port associated with the actuator connection port.

P

Physical signal port that controls the supply pressure.

S

Physical signal port that controls spool displacement.

Examples

The Closed-Loop Hydraulic Actuator Model for Real-Time Simulation example is an example of using this valve, along with other blocks optimized for real-time and HIL simulation.

See Also

4-Way Directional Valve

Introduced in R2010a

6-Way Directional Valve A

Configuration A of hydraulic continuous 6-way directional valve

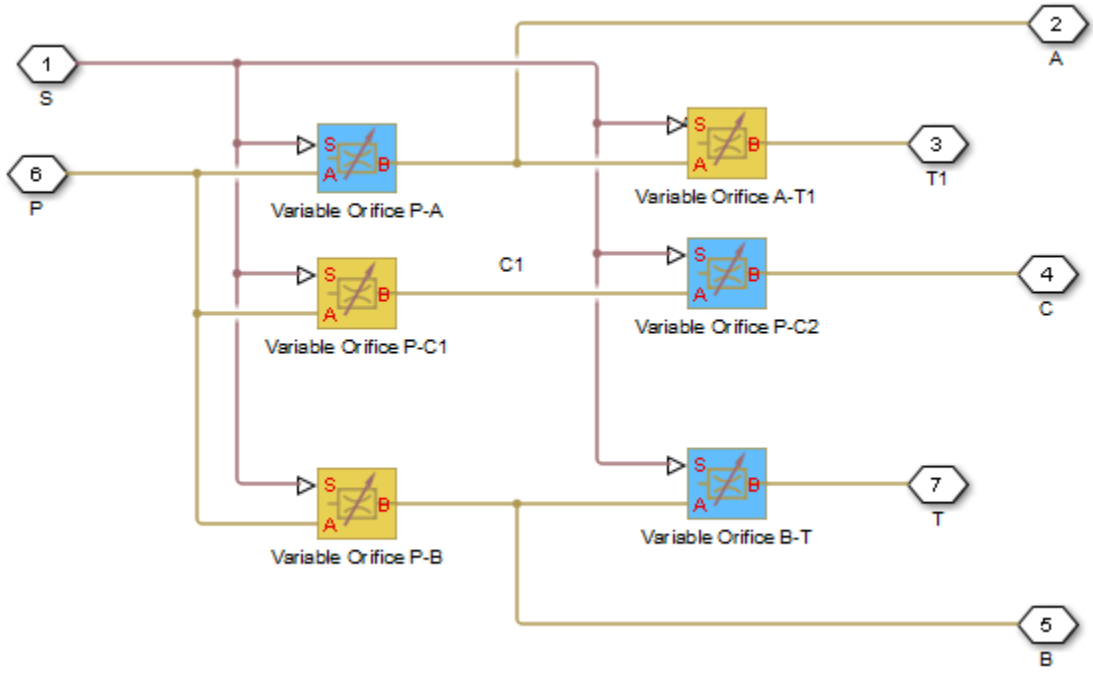
Library

Directional Valves

Description

The 6-Way Directional Valve A block simulates a configuration of hydraulic continuous 6-way directional valve with pump port P, two return ports T and T1, and three actuator ports A, B, and C. Use the valve for applications with two actuators, each being controlled by a valve of this type. When both valves are in neutral position, the pump is unloaded. If any of the valves is shifted from neutral, the diverting line is cut off and the respective actuator is fed at fuel pump pressure. For more details, see the Front-Loader Actuation System example, which uses a similar system of two valves.

The 6-Way Directional Valve A block has six hydraulic connections, corresponding to the valve hydraulic ports (A, B, C, P, T, and T1), and one physical signal port connection (S), which controls the spool position. The block is built of six Variable Orifice blocks, connected as shown in the following diagram.



All blocks are controlled by the same position signal, provided through the physical signal port S, but the **Orifice orientation** parameter in the block instances is set in such a way that positive signal at port S opens the orifices colored blue in the block diagram (orifices P-A, B-T, and P-C2) and closes the orifices colored yellow (orifices A-T1, P-B, and P-C1). As a result, the openings of the orifices are computed as follows:

$$h_{PA} = h_{PA0} + x$$

$$h_{PB} = h_{PB0} - x$$

$$h_{AT1} = h_{AT10} - x$$

$$h_{BT} = h_{BT0} + x$$

$$h_{PC1} = h_{PC10} - x$$

$$h_{PC2} = h_{PC20} + x$$

where

h_{PA}	Orifice opening for the `Variable Orifice P-A` block
h_{PB}	Orifice opening for the `Variable Orifice P-B` block
h_{AT1}	Orifice opening for the `Variable Orifice A-T1` block
h_{BT}	Orifice opening for the `Variable Orifice B-T` block
h_{PC1}	Orifice opening for the `Variable Orifice P-C1` block
h_{PC2}	Orifice opening for the `Variable Orifice P-C2` block
h_{PA0}	Initial opening for the `Variable Orifice P-A` block
h_{PB0}	Initial opening for the `Variable Orifice P-B` block
h_{AT10}	Initial opening for the `Variable Orifice A-T1` block
h_{BT0}	Initial opening for the `Variable Orifice B-T` block
h_{PC10}	Initial opening for the `Variable Orifice P-C1` block
h_{PC20}	Initial opening for the `Variable Orifice P-C2` block
x	Control member displacement from initial position

For information on the block parameterization options, basic parameter descriptions, assumptions and limitations, global and restricted parameters, see the 4-Way Directional Valve block reference page.

Parameters

- “Basic Parameters Tab” on page 1-262
- “Valve Opening Offsets Tab” on page 1-266

Basic Parameters Tab

Model parameterization

Select one of the following methods for specifying the valve:

- `By maximum area and opening` — Provide values for the maximum valve passage area and the maximum valve opening. The passage area is linearly

dependent on the control member displacement, that is, the valve is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- `By area vs. opening table` — Provide tabulated data of valve openings and corresponding valve passage areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of valve openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Valve passage maximum area

Specify the area of a fully opened valve. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-3` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Valve opening vector, s

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated valve openings** values will be used together with **Tabulated valve passage area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Valve passage area vector

Specify the vector of output values for valve passage area as a one-dimensional array. The valve passage area vector must be of the same size as the valve openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e +06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.

- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the valve, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Valve Opening Offsets Tab

Between ports P and A

Orifice opening of the **P-A** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and B

Orifice opening of the **P-B** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports A and T1

Orifice opening of the **A-T1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports B and T

Orifice opening of the **B-T** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `-2.5e-3` m.

Between ports P and C1

Orifice opening of the **P-C1** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Between ports P and C2

Orifice opening of the **P-C2** flow path at zero spool displacement. Specify a positive offset to model an underlapped valve or a negative offset to model an overlapped valve. The default value of `0` corresponds to a zero-lapped valve. The default value is `2.5e-3` m.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pressure supply line inlet.

`T`

Hydraulic conserving port associated with the first return line connection.

`T1`

Hydraulic conserving port associated with the second return line connection.

`A`

Hydraulic conserving port associated with the actuator connection port.

`B`

Hydraulic conserving port associated with the actuator connection port.

`C`

Hydraulic conserving port associated with the actuator connection port.

`S`

Physical signal port to control spool displacement.

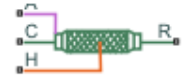
See Also

4-Way Directional Valve C | 4-Way Directional Valve H | 4-Way Directional Valve D | 4-Way Directional Valve K | 4-Way Directional Valve | 4-Way Directional Valve A | 4-Way Directional Valve B | 4-Way Directional Valve E | 4-Way Directional Valve F | 4-Way Directional Valve G

Introduced in R2009b

Air Muscle Actuator (G)

Linear actuator with force characteristics of biological muscle
Library: Simscape / Fluids / Gas / Actuators



Description

The Air Muscle Actuator (G) block models a linear actuator popular in robotics for its characteristics reminiscent of biological muscle. The actuator comprises an expandable bladder in a braided shell. When the bladder is pressurized, the pair widens and simultaneously shortens, producing at their end caps a contractile force. The bladder is pressurized at Gas port **A**; the force is exerted at Mechanical Translational ports **R** and **C**.

Air Muscle in Relaxed State

Air muscles are often installed in pairs—one muscle serving as agonist, the other as antagonist. Pairs of this sort are common in the human body, where the biceps (in the arm) accompanies the triceps, and the quadriceps (in the leg) accompanies the hamstrings. The muscles attach at one end to a joint, but at an offset so as to produce a torque. When the net torque is other than zero, and if loading conditions allow, the joint rotates.

Actuator Force

The mass and energy balances of the actuator are as described for the Translational Mechanical Converter (G) block. The actuator force, however, is based on the standard Chou-Hannaford equation (with two corrections made for the simplifying assumptions of the original model). In its original form, the Chou-Hannaford equation gives:

$$F_{\text{C-H}} = \frac{\pi D_{\text{M}}^2 P}{4}\left[3\left(\frac{L}{l}\right)^2 - 1\right],$$

where:

- F is the contractile force exerted by the actuator on its ends. The subscript `C-H` denotes the theoretical value of the original Chou-Hannaford model.
- D is the diameter of the bladder and shell assembly. The subscript `M` denotes its maximum theoretical value—that in which the braids of the shell are at right angles to its longitudinal axis.
- P is the gauge pressure in the bladder (measured against the environment external to the actuator).
- L is the length of the actuator (the distance between the mechanical ports **R** and **C**.
- l is the natural length of a braid (before it is stretched in a pressurized bladder). The braids, as they are wound about the longitudinal axis of the actuator, are always longer than the actuator itself).

The maximum theoretical diameter of the actuator is defined as:

$$D_{\text{M}} = \frac{l}{n\pi},$$

where n is the number of turns that a braid makes about the longitudinal axis of the actuator.

Implicit in the Chou-Hannaford equation are the assumptions of infinitely thin bladder and shell and of inelastic braids incapable of stretching. Both assumptions can lower the accuracy of the model and are corrected for in this block. The correction for the stretching in a braid replaces the constant length l with the variable length l^*:

$$l^* = \frac{Cl + \sqrt{(Cl)^2 + 12L^2(C+1)}}{2(C+1)} + \frac{2nPD^2}{Ed},$$

where l is the natural length of a braid used in the original Chou-Hannaford equation and:

- C is a correction term for braid stretching.

- E is Young's modulus of elasticity for the material of the braids.
- d is the diameter of a strand in a braid (each braid being a bundle of tightly intertwined strands).

The correction term for braid stretching is defined as:

$$C = \frac{n^2 \pi^2 E d^2 N}{P l L},$$

where N is the total number strands in the braided shell. The correction for the thickness of the bladder and shell adds to the total actuator force the factor:

$$F_{\mathrm{T}} = \pi P \left[t \left(2D - \frac{D_{\mathrm{M}}^2}{D} \right) - t^2 \right]$$

where t is the aggregate thickness of the bladder and shell and the subscript T denotes the correction for thickness. The total actuator force is:

$$F = F_{\mathrm{H\text{-}F}} + F_{\mathrm{T}},$$

where the strand length used in the calculation of the Chou-Hannaford term is the variable l^*. This force is counteracted at the limits of extension and contraction by translational hard stops. These are modeled as described for the Translational Hard Stop block.

Modeling Assumptions

- There is no flow resistance between the gas entrance (port **A**) and the interior of the actuator.
- There is no thermal resistance between the actuator wall (port **H**) and the gas that it encloses.
- The actuator is hermetic and does not leak.
- The effects of friction and inertia are ignored.
- The bladder and shell are perfectly cylindrical no matter their inflation level.
- The longitudinal elasticity of the bladder is ignored.

Ports

Conserving

`A` — Actuator inlet

gas

Gas opening for pressurizing the actuator. Connect this port to the pressure source for the actuator. The pressure at this port controls the force generated by the actuator.

`R` — Actuator end cap

mechanical translational

One of two end caps at which the actuator force is applied. Connect this port to the fixture on which the actuator is to be mounted or to the load that the actuator is to move.

`C` — Actuator end cap

mechanical translational

One of two end caps at which the actuator force is applied. Connect this port to the fixture on which the actuator is to be mounted or to the load that the actuator is to move.

`H` — Actuator wall

thermal

Thermal boundary condition between the gas volume of the actuator and its surroundings. Connect this port to blocks in the Thermal domain to capture heat transfer by conduction, convection, or radiation, for example, or to insulate the actuator from the environment.

Parameters

`Initial actuator length` — Distance between end caps when unloaded and unpressurized

`0.575 m` (default) | positive scalar in units of length

Distance between the end caps of the actuator in the relaxed state, when it is both unloaded and unpressurized. Inflating the actuator during simulation causes it to widen and simultaneously to shorten. The actuator is shorter than its braids, which is to say that its initial length should be less than the **Unstretched braid length** parameter.

`Unstretched braid length` — Length of straightened braid when unloaded
`0.65 m` (default) | positive scalar in units of length

Length of a braid as measured with its turns straightened and any loads removed. If the braids are elastic, their (common) length will vary with during simulation pressure and load. As the braids are wound about the longitudinal axis of the actuator, their length is always greater than the **Initial actuator length** parameter.

`Number of turns per braid` — Number of turns in a braid about the actuator length axis
`4.7` (default) | positive unitless scalar

Number of turns that a braid makes about the longitudinal axis of the actuator from one end cap to the second.

`Braid stretching` — Choice of whether to model the stretching of a braid
`Inelastic braids` (default) | `Elastic braids`

Choice of whether to model the stretching of a braid. The default option, `Inelastic braids`, treats the braids as flexible but inextensible. This is the assumption implicit in the original Chou-Hannaford equation. The length of a braid is then constant during simulation.

The alternative option, `Elastic braids`, allows the braids to extend with actuator load and pressure. The length of the braid is then treated as variable during simulation. This change adds a correction to the Chou-Hannaford equation for force.

`Thickness of braided shell and bladder` — Combined thickness of shell and bladder
`0.075 m` (default) | positive scalar in units of length

Sum of the thicknesses of the braided shell and of the bladder inside it. A finite thickness adds to the Chou-Hannaford equation for force a correction term. (In the original Chou-Hannaford equation, the shell and bladder are assumed to be infinitely thin.)

`Cross-sectional area at port A` — Area normal to flow at the gas port
`0.01 m` (default) | positive scalar in units of length

Area normal to flow at the gas port of the actuator.

`Total number of braids` — Number of braids in the braided shell
`80` (default) | positive unitless scalar

Number of braids in the shell surrounding the bladder. As the braids are each a bundle of intertwined strands, the number of braids is not generally the same as the number of strands.

Dependencies

This parameter is active when the **Braid stretching** option is set to `Elastic Braids`.

`Number of strands per braid` — Number of individual strands in a braid
`3` (default) | positive unitless scalar

Number of strands in a braid. The product of this number and the **Total number of braids** parameter gives the total number of strands in the shell.

Dependencies

This parameter is active when the **Braid stretching** option is set to `Elastic Braids`.

`Strand diameter` — Diameter of a strand
`0.25 m` (default) | positive scalar in units of length

Average diameter of a strand throughout its length.

Dependencies

This parameter is active when the **Braid stretching** option is set to `Elastic Braids`.

`Young's modulus of strands` — Degree to which the strands extend under pressure
`3e9 GPA` (default)

Young's modulus of elasticity for the material of the strands. This parameter determines the degree to which the strands extend under pressure.

Dependencies

This parameter is active when the **Braid stretching** option is set to `Elastic Braids`.

`Environment pressure specification` — Pressure condition outside of the actuator
`Atmospheric pressure` (default) | `Specified pressure`

Pressure condition in the immediate surroundings of the actuator. The gauge pressure in the actuator is obtained relative to this pressure. Select `Atmospheric pressure` to use

the atmospheric value specified in the Two-Phase Fluid Properties (2P) block or `Specified pressure` to use a different pressure.

`Environment pressure` — Absolute pressure outside of the actuator
`0.101325` (default) | scalar in units of pressure

Absolute pressure in the immediate surroundings of the actuator. The actuator contracts when the gauge pressure at port **A** (measured against this value) is greater than zero.

Dependencies

This parameter is active when the **Environment pressure specification** option is set to `Specified pressure`.

`Contact stiffness of braids at full muscle contraction or extension` — Stiffness coefficient for contact force at actuator motion limits
`1e6 N/m` (default)

Stiffness coefficient for the contact force at the limits of the actuator's range of motion. The contact force keeps the actuator from extending or contracting beyond its physical limits. The contact force is modeled using a spring-damper system. The spring element adds a restorative force that pulls the end caps back once they reach either limit of motion.

For more information on the force model, see the Translational Hard Stop block.

`Contact damping of braids at full muscle contraction or extension` — Damping coefficient for contact force at actuator motion limits
`1e4 N/(m/s)` (default)

Damping coefficient for the contact force at the limits of the actuator's range of motion. The contact force keeps the actuator from extending or contracting beyond its physical limits. This force is modeled using a spring-damper system. The damper adds a viscous force to the end caps once they reach a limit of motion, causing them to slow to a stop.

For more information on the force model, see the Translational Hard Stop.

See Also

Fan (G) | Translational Hard Stop | Translational Mechanical Converter (G)

Introduced in R2018b

Angle Sensor

Ideal angle sensor with measuring range from 0 to 360 degrees

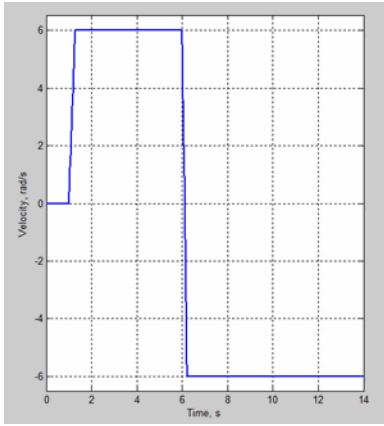
Library

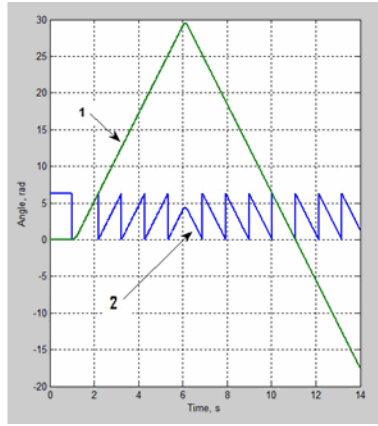
Pumps and Motors

Description

The Angle Sensor block represents an ideal angle sensor with measuring range from 0 to 360 degrees. To measure an angular displacement, the sensor must be connected to a mechanical rotational port. No load is imposed on the measured object. The sensor output is kept within the range from 0 to 2π radians (360 degrees) regardless of the number of revolutions performed by the object and the direction of rotation. The sensor is ideal because it does not account for inertia, friction, and so on.

The difference between the Angle Sensor block and the Ideal Rotational Motion Sensor block, available in the Simscape Foundation library, is demonstrated in the following figure.





In this example, the object is moving at 6 rad/s in the positive direction for the first 5 seconds, and then switches to the negative direction at the same speed. The Ideal Rotational Motion Sensor block output (line 1) shows that the object turned forward by 30 rad and then turned back in the negative direction, continuing until –20 rad. The reading of the Angle Sensor block (line 2) stays in the range from 0 to 2π rad.

Availability of the Angle Sensor block considerably simplifies development of models with complex relationship between model parameters and rotation angle, such as pumps and motors.

Basic Assumptions and Limitations

- The sensor is assumed to be ideal. No load is imposed on the object.
- No inertial effects are considered.

Parameters

Phase angle

This parameter lets you account for angular shift between the object and the sensor input. The default value is `0`.

Ports

The block has the following ports:

`W`

Mechanical rotational conserving port associated with the probe of the sensor.

`A`

Physical signal port that outputs the angular displacement of the object.

Examples

The Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control example models a test rig designed to investigate interaction between an axial-piston pump and a typical control unit, simultaneously performing the load-sensing and pressure-limiting functions. To assure required accuracy, the model of the pump must account for such features as interaction between pistons, swash plate, and porting plate, which makes it necessary to build a detailed pump model.

The Angle Sensor block is used in each of the piston model subsystems (`Piston_1`, ..., `Piston_5`), where it provides the angular position of the respective piston as input to the Porting Plate Variable Orifice blocks.

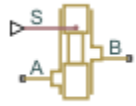
See Also

Ideal Rotational Motion Sensor

Introduced in R2011a

Annular Orifice

Hydraulic variable orifice created by circular tube and round insert

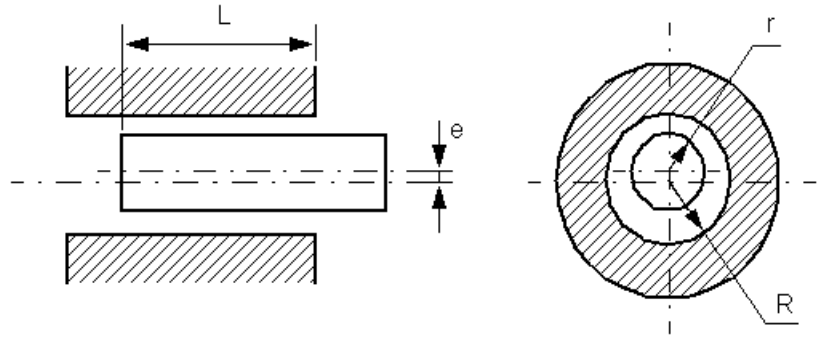


Library

Orifices

Description

The Annular Orifice block models a variable orifice created by a circular tube and a round insert, which may be eccentrically located with respect to the tube. The radial gap between the tube and the insert and its axial length are assumed to be essentially smaller than the insert diameter, causing the flow regime to be laminar all the time. A schematic representation of the annular orifice is shown in the following illustration.



The flow rate is computed using the Hagen-Poiseuille equation (see [1] on page 1-282):

$$q = \frac{\pi R(R-r)^3}{6\nu\rho L} \bullet \left(1 + \frac{3}{2}\varepsilon^2\right) \bullet p$$

$$\varepsilon = \frac{e}{R-r}$$

where

q	Flow rate
p	Pressure differential
R	Orifice radius
r	Insert radius
L	Overlap length
ε	Eccentricity ratio
e	Eccentricity
ρ	Fluid density
ν	Fluid kinematic viscosity

Use this block to simulate leakage path in plungers, valves, and cylinders.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as

$\Delta p = p_A - p_B$, . Positive signal at the physical signal port S increases or decreases the overlap, depending on the value of the parameter **Orifice orientation**.

Basic Assumptions and Limitations

Fluid inertia is not taken into account.

Parameters

Orifice radius

The radius of the tube. The default value is `0.01` m.

Insert radius

The radius of the insert. The default value is `0.0098` m.

Eccentricity

The distance between the central axes of the insert and the tube. The parameter can be a positive value, smaller than the difference between the radius of the tube and the radius of the insert, or equal to zero for coaxial configuration. The default value is `0`.

Initial length

Initial overlap between the tube and the insert. The parameter must be positive. The value of initial length does not depend on the orifice orientation. The default value is `0.003` m.

Orifice orientation

The parameter is introduced to specify the effect of the control signal on the orifice overlap. The parameter can be set to one of two options: `Positive signal increases overlap` or `Negative signal increases overlap`. The default value is `Positive signal increases overlap`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Orifice orientation**

All other block parameters are available for modification.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port that controls the insert displacement.

References

[1] Noah D. Manring, *Hydraulic Control Systems*, John Wiley & Sons, 2005

See Also

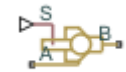
Constant Area Hydraulic Orifice | Fixed Orifice | Orifice with Variable Area Round Holes | Orifice with Variable Area Slot | Variable Area Hydraulic Orifice | Variable Orifice

Introduced in R2006b

Ball Valve

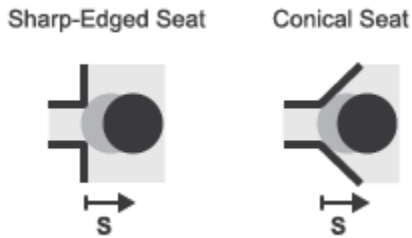
Valve with a sliding ball control mechanism

Library: Simscape / Fluids / Hydraulics (Isothermal) / Valves / Flow Control Valves



Description

The Ball Valve block models an orifice with a variable opening area controlled by a sliding ball mechanism. The opening area changes with the relative position of the ball—the *valve lift*. A displacement toward the orifice decreases flow while a displacement away increases flow. The interface between the orifice and the ball—the *valve seat*—can be `Sharp-edged`, shown left in the figure, or `Conical`, shown right.



Ball Valve Seat Types

Opening Area

The valve lift is a function of the displacement signal specified through port **S**. The two can, but generally do not, have the same value. The valve lift differs from the displacement whenever the **Ball displacement offset** parameter is nonzero:

$$h(x) = x_0 + s,$$

where:

- h is the valve lift.
- x_0 is the ball displacement offset.
- s is the ball displacement (relative to the specified offset).

The valve is fully closed when the valve lift is equal to zero or less. It is fully open when the valve lift reaches or exceeds a (geometry-dependent) value sufficient to completely clear the orifice. A fully closed valve has an opening area equal to the specified **Leakage area** parameter while a fully open valve has the maximum possible opening area. Adjusting for internal leakage:

$$A_{Max} = \pi r_O^2 + A_{Leak},$$

where:

- A_{Max} is the maximum opening area.
- r_{O} is the orifice radius.
- A_{Leak} is the internal leakage area between the ports.

At intermediate values of the valve lift, the opening area depends on the valve seat geometry. If the **Valve seat specification** parameter is set to `Sharp-Edged`, the opening area as a function of valve lift is:

$$A(h) = \pi r_O \left(1 - \left[\frac{r_B}{d_{OB}}\right]^2\right) d_{OB}(h),$$

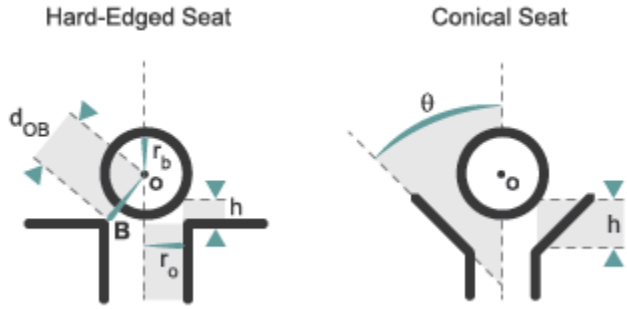
where:

- A is the opening area at a given valve lift value.
- r_{B} is the ball radius.
- $d_{\mathrm{OB}}(h)$ is the distance from the center of the ball (point **O** in the figure) to the edge of the orifice (point **B**). This distance is a function of the valve lift (h).

If the **Valve seat specification** parameter is set to `Conical`, the opening area becomes:

$$A(h) = \pi\, h\, cos(\theta) \sin(\theta) \left(d_{OB}(h) + h \sin(\theta)\right),$$

where θ is the angle between the conical surface and the orifice centerline. The geometrical parameters and variables used in the equations are shown in the figure.



Valve Geometries

Flow Rate

The volumetric flow rate through the valve is a function of the opening area, *A(h)*, and of the pressure differential between the valve ports:

$$q = C_D A(h)\sqrt{\frac{2}{\rho}}\frac{\Delta p}{\left((\Delta p)^2 + p_{Cr}^2\right)^{1/4}},$$

where:

- C_D is the flow discharge coefficient.
- ρ is the hydraulic fluid density.
- Δp is the pressure differential between the valve ports, defined as:

  $$\Delta p = p_A - p_B,$$

  where p_A is the pressure at port **A** and p_B is the pressure at port **B**.
- p_{Cr} is the minimum pressure required for turbulent flow.

The critical pressure p_{Cr} is computed from the critical Reynolds number as:

$$p_{Cr} = \frac{\rho}{2}\left(\frac{Re_{Cr}\nu}{C_D D_H}\right)^2,$$

where:

- Re_{Cr} is the critical Reynolds number.
- ν (nu) is the hydraulic fluid dynamic viscosity.
- D_{H} is the orifice hydraulic diameter:

$$D_H = \begin{cases} D_H^{Min}, & \text{if } h \le 0 \\ D_H^{Max} + D_H^{Min}, & \text{if } h \ge h_{Max}, \\ \dfrac{4A}{l} + D_H^{Min}, & \text{otherwise} \end{cases}$$

in which:

- $D_{\mathrm{H}}^{\mathrm{Min}}$ is the minimum hydraulic diameter, corresponding to the smallest attainable flow area, the leakage flow area.
- $D_{\mathrm{H}}^{\mathrm{Max}}$ is the maximum hydraulic diameter, corresponding to the largest attainable flow area, that of the valve in the fully open position.
- l is the wetted length of the valve perimeter—which can, but need not, be that of a circle.

Assumptions

Fluid inertia is assumed to be negligible.

Ports

Input

S — Ball displacement, `m`

physical signal

Physical signal input port for the ball displacement.

Conserving

`A` — Valve inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port associated with the valve inlet.

`B` — Valve outlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port associated with the valve outlet.

Parameters

Parameters Tab

`Valve seat specification` — Valve seat geometry type
`Sharp-edged` (default) | `Conical`

Valve seat geometry type. The seat geometry affects the opening area equations used during simulation.

`Ball diameter` — Diameter of the sliding ball
`0.01` m (default) | scalar in units of length

Diameter of the ball used to control the valve opening area. This parameter must be greater than the **Orifice diameter** value.

`Orifice diameter` — Diameter of the valve orifice
`0.005` m (default) | scalar in units of length

Diameter of the valve orifice. This parameter must be smaller than the **Ball diameter** value.

`Ball displacement offset` — Gate lift at a zero displacement signal
`0` m (default) | scalar in units of length

Gate lift *h* corresponding to a zero displacement signal. This parameter must be greater than or equal to zero.

`Discharge coefficient` — Semi-empirical parameter for valve capacity characterization
`0.7` (default) | unitless scalar between 0 and 1

Ratio of the actual and theoretical flow rates through the valve. This parameter depends on the geometrical properties of the valve. Values are usually provided in textbooks and manufacturer data sheets.

`Leakage area` — Internal leakage area between valve inlets
`1e-12 m^2` (default) | positive scalar in units of area

Total area of internal leaks in the completely closed state. The purpose of this parameter is to maintain the numerical integrity of the fluid network by preventing a portion of that network from becoming isolated when the valve is completely closed.

`Laminar transition specification` — Choice of parameter used to define the laminar-turbulent transition
`Pressure ratio` (default) | `Reynolds number`

Select the parameter to base the laminar-turbulent transition on. Options include:

- `Pressure ratio` — Flow transitions between laminar and turbulent at the pressure ratio specified in the **Laminar flow pressure ratio** parameter. Use this option for the smoothest and most numerically robust flow transitions.
- `Reynolds number` — The transition occurs at the Reynolds number specified in the **Critical Reynolds number** parameter. Flow transitions are more abrupt and can cause simulation issues at near-zero flow rates.

`Laminar flow pressure ratio` — Pressure ratio at the laminar-turbulent flow transition
`0.999` (default) | unitless scalar between 0 and 1

Pressure ratio at which the flow transitions between the laminar and turbulent regimes. The pressure ratio is the fraction of the outlet pressure over the inlet pressure.

Dependencies

This parameter is active when the **Laminar transition specification** parameter is set to `Reynolds number`.

`Critical Reynolds number` — Reynolds number at the laminar-turbulent flow transition
`12` (default) | unitless positive scalar

Maximum Reynolds number for laminar flow. This parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value, 12, corresponds to a round orifice in thin material with sharp edges.

Dependencies

This parameter is active when the **Laminar transition specification** parameter is set to `Reynolds number`.

Variables Tab

`Flow Rate` — Volumetric flow rate through the valve at time zero

`0.001` m^3/s (default) | scalar number in units of volume per unit time

Volumetric flow rate through the valve at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Pressure drop` — Pressure drop from port A to port B at time zero

`0` Pa (default) | scalar number in units of pressure

Pressure drop from port A to port B at time zero. The pressure drop is positive if pressure is greater at port A than at port B. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

See Also

Gate Valve | Needle Valve | Poppet Valve | Pressure-Compensated Flow Control Valve

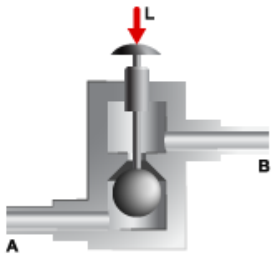
Ball Valve (G)

Valve with longitudinally translating ball as control element

Library: Simscape / Fluids / Gas / Valves & Orifices / Flow Control Valves

Description

The Ball Valve (G) block models an orifice with a translating ball (technically a *poppet*) as a flow control mechanism. The ball is sized to fully cover the orifice and arranged to shift in line with the flow. Its seat is perforated, with the opening, part of the orifice through which the flow must pass, being either sharp-edged or conical in shape. The distance of the ball to the seat determines the opening area of the valve.



Open ball valve with conical seat

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Ball valves (generally) lack the opening characteristics to precisely modulate or throttle flow rate; they more commonly serve as shutoff and isolation valves, binary on/off switches which are often either fully open or maximally closed.

Ball Mechanics

In a real valve, the ball often connects by a stem to a push button. When the button is pressed—by the magnetic force of a solenoid coil, say, or by the cyclical action of a rotary cam—the ball shifts from its seat, progressively opening the valve up to a maximum. A spring between the ball and the body of the valve acts as a return mechanism, allowing the ball to revert to its normal position once the button is released.

The block captures the motion of the ball but not the detail of its mechanics. The motion derives from a normalized displacement specified as a physical signal at port **L**. The normalization is with respect to the maximum position of the ball (at which the valve is fully open). It helps to think of displacement, position, and related quantities as fractions (normally from `0` to `1`) rather than as lengths.

If the action of the push button and return spring matter in your model, you can capture these elements separately using other Simscape blocks. A Simscape Mechanical subsystem makes a good source for the ball displacement signal. In many cases, however, it suffices to know what displacement to impart to the ball. You can then usually ignore the mechanics of the valve.

Ball Position

The displacement signal allows the block to compute the instantaneous position of the ball, from which the opening of the valve follows. The opening is easily understood as a flow area but, for ease of modeling, it is often best expressed as a flow coefficient or sonic conductance. (The ``ease of modeling'' depends on the data available from the manufacturer.)

The position and displacement variables measure different things. The (instantaneous) position gives the distance of the ball to its resting place on the seat;

the displacement gives only that distance to its normal (unactuated) position. The normal position, a fixed coordinate, need not be zero: the ball can be installed so that it is normally off the seat. (The valve is then partially open even when it is disconnected and its poppet at rest.)

The normal distance between the ball and its seat is the valve lift control offset, specified in the block parameter of the same name. Think of its as the permanent displacement given to the ball while assembling the valve. The variable displacement from port **L**, on the other hand, captures the motion of the ball during operation of the valve, after it has

been assembled and installed. The instantaneous position of the ball is the sum of the two:

$$h(L) = L + h_0,$$

where:

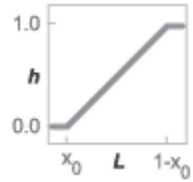
- h is the instantaneous position of the ball, normalized against its maximum value. This variable can range from `0` to `1`, with `0` giving a maximally closed valve and `1` a fully open valve. If the calculation should return a number outside of this range, that number is set to the nearest bound (`0` if the result is negative, `1` otherwise). In other words, the normalized position *saturates* at `0` and `1`.
- L is the variable displacement of the ball, normalized against the maximum position of the same. This variable is obtained from the physical signal at port **L**. There are no restrictions on its value. You can make it smaller than `0` or greater than `1`, for example, to compensate for an equally extreme valve offset.
- h_0 is the fixed offset of the ball relative to its seat in the normal position (when the valve is disconnected and free of inputs). Its value too is normalized against the maximum position of the ball, though there is no requirement that it lie between `0` and `1`.

Numerical Smoothing

The normalized position, h, spans three regions. At a sufficiently small displacement, it saturates at `0` and the valve is fully closed. At a sufficiently large displacement, it saturates at `1` and the valve is fully open. In between, it varies linearly between its saturation bounds, giving a valve that is partially open.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish the simulation run, perhaps excessively so for practical use in real-time simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized position with sharp transitions

To remove the slope discontinuities, the block smoothes them over a small portion of the opening curve. The smoothing, which adds a slight distortion at each transition, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_{\mathrm{L}} = 3\bar{h}_{\mathrm{L}}^{2} - 2\bar{h}_{\mathrm{L}}^{3}$$

and

$$\lambda_{\mathrm{R}} = 3\bar{h}_{\mathrm{R}}^{2} - 2\bar{h}_{\mathrm{R}}^{3},$$

where

$$\bar{h}_{\mathrm{L}} = \frac{h}{\Delta h^{*}}$$

and

$$\bar{h}_{\mathrm{R}} = \frac{h - \left(1 - \Delta h^{*}\right)}{\Delta h^{*}}.$$

In the equations:

- λ_{L} is the smoothing expression for the transition from the maximally closed position.
- λ_{R} is the smoothing expression for the transition from the fully open position.
- Δp^{*} is the (unitless) characteristic width of the smoothing region:

$$\Delta h^* = \frac{1}{2} f^*,$$

where f^* is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

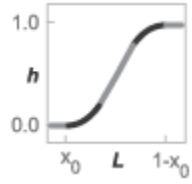
When the smoothing factor is `0`, the normalized ball position stays in its original form —no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the ball's travel range (with the normalized ball position taking the shape of an *S*-curve).

At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the travel range on each side (for a total smooth region of half that range).

The smoothing adds two new regions to the normalized ball position function—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$h^* = \begin{cases} 0, & h \le 0 \\ h\lambda_{\mathrm{L}}, & h < \Delta h^* \\ h, & h \le 1 - \Delta h^* \\ h\left(1 - \lambda_{\mathrm{R}}\right) + \lambda_{\mathrm{R}}, & h < 1 \\ 1 & h \ge 1 \end{cases},$$

where h^* is the smoothed valve opening area. The figure shows the effect of smoothing on the sharpness of the transitions.



Opening Area

The opening area at a given ball position depends in part on the geometry of its seat. That geometry, specified in the **Valve seat geometry** parameter, can be either `Sharp-edged` or `Conical`. In each case, the bottleneck region through which the flow must pass is in the shape of a conical frustum. The lateral surface of the frustum gives the opening area.

The frustum of a cone is any section of the same taken between two planes parallel to the base. In this block, the (imaginary) cone spans from the surface of the seat (where it has its base) to the center of the ball (its apex). The frustum is formed by cutting the cone where it intersects the ball.

The base of the cone, to be precise, is the ring of contact between the seat and the ball when the valve is in the closed position. The ring coincides with the edge of the seat only if it is sharp-edged. It lies inward from the edge if the seat is conical—somewhere along the conical surface between the orifice and the chamber of the ball.

A frustum of base radius R, top radius r, and slant height l has for its lateral surface area:

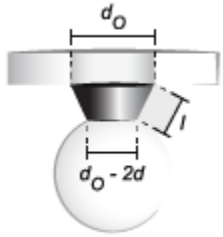
$$S = \pi(R + r)l.$$

To account for leakage flow, the block adds a small constant to this expression:

$$S = \pi(R + r)l + S_{\mathrm{Leak}},$$

where S_{Leak} is the small opening area that remains in the maximally closed valve due, for example, to valve bore tolerances, surface defects, or an imperfect seal between the ball and its seat. This area is obtained from the **Leakage area** block parameter.

If the seat is sharp-edged, the radius of the base of the frustum is that of the orifice itself (or half of the orifice diameter, D_{O}, specified in the block). The radius of the top face is a small decrement, δ, from that of the base. The decrement and the slant height, l, are functions of the normalized position of the ball (h). These lengths are shown in the figure.



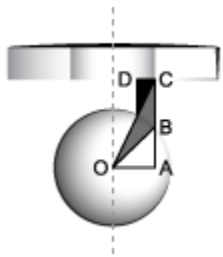
In terms of the orifice diameter, radius decrement, and slant height, the frustum area (S) becomes:

$$S = \pi\left[D_{\mathrm{O}} + \left(D_{\mathrm{O}} - 2\delta\right)\right]l + S_{\mathrm{Leak}}$$

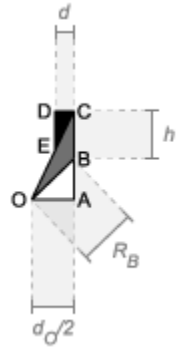
Rearranged and written in terms of the orifice radius ($2R_{\mathrm{O}} = D_{\mathrm{O}}$):

$$S = \pi\left(4R_{\mathrm{O}} - 2\delta\right)l + S_{\mathrm{Leak}}$$

The values of δ and l are obtained from expressions of trigonometry and proportion. These are based on the sides of the triangles shown in the figure. $\overline{CD}$ corresponds to the radius decrement (δ) and $\overline{CE}$ to the slant height (l). $\overline{OA}$ is equal in length to the orifice radius.



Other relevant lengths include $\overline{OE}$ (half the diameter of the ball, R_{B}) and $\overline{CB}$ (the position of the same, h). The various lengths are more clearly shown in the figure below. The diameters of the orifice (R_{O}) and of the ball (R_{B}) are obtained from the block parameters named for them. The ball position is calculated from the signal at port **L**.



The slant height can now be expressed as:

$$l = OC - R_{\mathrm{B}},$$

where *OC* is the distance from the center of the ball (point **O**) to a point on the edge of the orifice (**C**). From Pythagoras' theorem:

$$OC = \sqrt{\left[h\rho + AB\right]^2 + R_{\mathrm{O}}^2},$$

where ρ is the maximum ball position in units of length (in contrast to the normalized ball position, which is unitless). Its value is computed by solving the equality:

$$S(h) = S_{\mathrm{Max}},$$

where S_{Max} is the maximum opening area:

$$S_{\mathrm{Max}} = \pi R_{\mathrm{O}}^2,$$

where R_{O} is the radius of the orifice (half the value specified in the **Orifice diameter** block parameter.

Similarly for length *AB*:

$$AB = \sqrt{R_{\mathrm{B}}^2 - R_{\mathrm{O}}^2}$$

The radius decrement, δ, can likewise be expressed as:

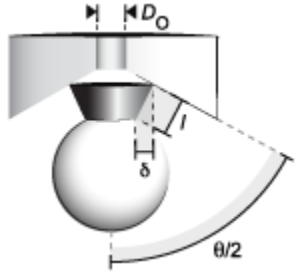
$$\delta = \frac{l}{OC} R_{\mathrm{O}}.$$

Combining these expressions and rearranging them, gives:

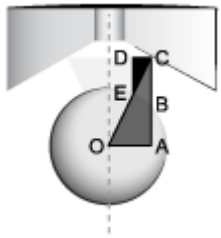
$$S = 2\pi R_{\mathrm{O}} OC \left[1 - \left(\frac{R_{\mathrm{B}}}{OC} \right)^2 \right] + S_{\mathrm{Leak}},$$

where *OC* is as defined above.

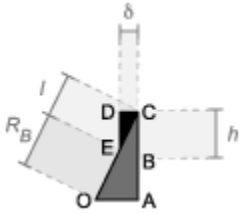
If the seat is conical, the ball comes to rest not on the rim of the orifice but on the conical surface that extends from it. The base of the frustum no longer matches the orifice in its radius and this variable is instead calculated from other lengths. The figure shows the frustum-shaped opening of the valve and the mismatch in size between its base and the orifice.



The calculations of the frustum dimensions are based on the triangles shown below. As in the sharp-edged calculations, $\overline{CD}$ is the decrement in radius from base to top (δ) and $\overline{CE}$ is the slant height of the frustum (l). The length of $\overline{OA}$ gives the base radius.



Other relevant dimensions include the cone angle (θ, taken between the sides of a cross section of the cone) and the ball radius (R_B). These and other dimensions are more clearly shown in the figure below.



In terms of these dimensions, the lateral surface area of the frustum is:

$$S = \pi\left(OA - 2\delta\right)l + S_{\text{Leak}},$$

where OA is the radius of the base:

$$OA = \left(l + R_B\right)\cos\left(\frac{\theta}{2}\right),$$

where θ is the angle between the side of the conical seat and its center line, obtained from the **Cone angle** parameter (specific to the `Conical` parameterization). The slant height of the frustum is:

$$l = \sin\left(\frac{\theta}{2}\right)h\rho,$$

where, as in the sharp-edged calculations, ρ is the maximum position of the ball in units of length. The decrement in radius from base to top is:

$$\delta = \cos\left(\frac{\theta}{2}\right) l.$$

Combining these expressions, applying basic trigonometric relationships for double angles, and arranging terms gives for the opening area:

$$S = 2\pi R_B h\rho \sin(\theta) + \pi h^2 \rho^2 \sin(\theta)\sin\left(\frac{\theta}{2}\right) + S_{Leak},$$

Sonic Conductance

As the opening area varies during simulation, so does the mass flow rate through the valve. The relationship between the two variables, however, is indirect. The mass flow rate is defined in terms of the valve's sonic conductance and it is this quantity that the opening area truly determines.

Sonic conductance, if you are unfamiliar with it, describes the ease with which a gas will flow when it is choked—when its velocity is at its theoretical maximum (the local speed of sound). Its measurement and calculation are covered in detail in the ISO 6358 standard (on which this block is based).

Only one value is commonly reported in valve data sheets: one taken at steady state in the fully open position. This is the same specified in the block dialog box (when the **Valve parameterization** setting is `Sonic conductance`). For values across the opening range of the valve, this maximum is scaled by the (normalized) valve opening area:

$$C(S) = \frac{S}{S_{Max}} C_{Max},$$

where C is sonic conductance and the subscript `Max` denotes the specified (manufacturer's) value. The sonic conductance varies linearly between C_{Max} in the fully open position and $S_{Leak} \div S_{Max} \times C_{Max}$ in the maximally closed position—a value close to zero and due only to internal leakage between the ports.

Because sonic conductance may not be available (or the most convenient choice for your model), the block provides several equivalent parameterizations. Use the **Valve parameterization** drop-down list to select the best for the data at hand. The parameterizations are:

- `Compute from geometry`
- `Sonic conductance`
- `Cv coefficient (USCS)`
- `Kv coefficient (SI)`

The parameterizations differ only in the data that they require of you. Their mass flow rate calculations are still based on sonic conductance. If you select a parameterization other than `Sonic conductance`, then the block converts the alternate data—the (computed) opening area or a (specified) flow coefficient—into an equivalent sonic conductance.

The flow coefficients measure what is, at bottom, the same quantity—the flow rate through the valve at some agreed-upon temperature and pressure differential. They differ only in the standard conditions used in their definition and in the physical units used in their expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Cv coefficient (USCS)`.
- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `1 bar`; it is expressed in metric units of m^3/h. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Kv coefficient (SI)`.

If the valve parameterization is set to `Cv Coefficient (USCS)`, the sonic conductance is computed at the maximally closed and fully open valve positions from the **Cv coefficient (SI) at maximum flow** and **Cv coefficient (SI) at leakage flow** block parameters:

$$C = \left(4 \times 10^{-8} C_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where C_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`. (These are used in the mass flow rate calculations given in the Momentum Balance section.)

If the `Kv coefficient (SI)` parameterization is used instead, the sonic conductance is computed at the same valve positions (maximally closed and fully open) from the **Kv**

coefficient (USCS) at maximum flow and **Kv coefficient (USCS) at leakage flow** block parameters:

$$C = \left(4.758 \times 10^{-8} K_{\mathrm{v}}\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where K_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`.

For the `Restriction area` parameterization, the sonic conductance is computed (at the same valve positions) from the **Maximum opening area**, and **Leakage area** block parameters:

$$C = \left(0.128 \times 4S_{\mathrm{R}} / \pi\right) \mathrm{L}/(\mathrm{s\ bar}),$$

where S_R is the opening area at maximum or leakage flow. The subsonic index, m, is set to `0.5` while the critical pressure ratio, b_{cr} is computed from the expression:

$$0.41 + 0.272\left(\frac{S_{\mathrm{R}}}{S_P}\right)^{0.25},$$

where the subscript `P` refers to the inlet of the connecting pipe.

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is assumed to reflect entirely in the sonic conductance of the valve (or in the data of the alternate valve parameterizations).

When the flow is choked, the mass flow rate is a function of the sonic conductance of the valve and of the thermodynamic conditions (pressure and temperature) established at the inlet. The function is linear with respect to pressure:

$$\dot{m}_{\mathrm{ch}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}},$$

where:

- C is the sonic conductance inside the valve. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- ρ is the gas density, here at standard conditions (subscript `0`), obtained from the **Reference density** block parameter.
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (`in`) or at standard conditions (`0`), the latter obtained from the **Reference temperature** block parameter.

When the flow is subsonic, and therefore no longer choked, the mass flow rate becomes a nonlinear function of pressure—both that at the inlet as well as the reduced value at the outlet. In the turbulent flow regime (with the outlet pressure contained in the back-pressure ratio of the valve), the mass flow rate expression is:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{p_{\mathrm{r}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m ,$$

where:

- p_{r} is the back-pressure ratio, or that between the outlet pressure (p_{out}) and the inlet pressure (p_{in}):

$$P_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the flow becomes choked. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

When the flow is laminar (and still subsonic), the mass flow rate expression changes to:

$$\dot{m}_{\text{lam}} = C\rho_0 p_{\text{in}} \left[\frac{1-p_{\text{r}}}{1-b_{\text{lam}}}\right] \sqrt{\frac{T_0}{T_{\text{in}}}} \left[1-\left(\frac{b_{\text{lam}}-b_{\text{cr}}}{1-b_{\text{cr}}}\right)^2\right]^m$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes (obtained from the **Laminar flow pressure ratio** block parameter). Combining the mass flow rate expressions into a single (piecewise) function, gives:

$$\dot{m} = \begin{cases} \dot{m}_{\text{lam}}, & b_{\text{lam}} \le p_{\text{r}} < 1 \\ \dot{m}_{\text{tur}}, & b_{\text{cr}} \le p_{\text{r}} < p_{\text{lam}} \\ \dot{m}_{\text{ch}}, & p_{\text{r}} < b_{\text{Cr}} \end{cases},$$

with the top row corresponding to subsonic and laminar flow, the middle row to subsonic and turbulent flow, and the bottom row to choked (and therefore sonic) flow.

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of gas can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**. Note that in this block the flow can reach but not exceed sonic speeds.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the gas and the wall that surrounds it. No work is done on or by the gas as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\text{A}} + \phi_{\text{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Input

`L` — Valve control signal, `unitless`

physical signal

Input port through which to specify the control signal for the valve ball displacement. The ball position, or *valve lift*, is calculated as the sum of this signal and the valve lift control offset (specified in the block parameter of the same name).

When the ball position is `0`, the valve is maximally closed and flow is limited to internal leakage between the ports. When the ball position is `1`, the valve is fully open and flow is at the maximum allowed by the pressure drop between the ports.

Conserving

`A` — Valve entrance

gas

Opening through which the working fluid can enter or exit the valve. The direction of flow depends on the pressure differential established across the valve. Both forward and backward directions are allowed.

`B` — Valve entrance

gas

Opening through which the working fluid can enter or exit the valve. The direction of flow depends on the pressure differential established across the valve. Both forward and backward directions are allowed.

Parameters

`Valve seat specification` — Geometry of the entrance on which the ball must sit
`Sharp-edged` (default) | `Conical`

Geometry of the seat of the ball. The geometry selection determines which calculation to use for the opening area of the valve. See the block description for detail on the block calculations.

`Cone angle` — Angle between the conical seat of the valve and its center lines
`120 deg` (default) | positive scalar in angular units

Angle formed by the slope of the conical seat against its center line.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve seat specification** parameter is set to `Conical`.

`Ball diameter` — Diameter of the ball used to regulate the valve opening
`0.01 m` (default) | scalar in units of length

Diameter of the ball control element.

`Orifice diameter` — Diameter of the orifice that the ball serves to constrict
`7e-3 m` (default) | scalar in units of length

Diameter of the orifice of the valve. In those with a conical seat, the diameter is that of the root of the seat (where it meets the orifice). The orifice is assumed to be constant in cross section throughout its length (from the seat to the opposite entrance).

`Valve lift control offset` — Distance of the ball from its seat when the valve is in the normal position
`0` (default) | unitless scalar

Displacement of the ball from the root of its seat, expressed as a fraction of its maximum travel distance, when no control input is provided or when that input is zero (the *normal* valve position. The instantaneous ball position is calculated during simulation as the sum of this offset and the control signal specified at port **L**. The valve is partially open in its normal position when the offset is a fraction between `0` and `1`.

`Valve parameterization` — Method by which to characterize the opening of the valve
`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the **Sonic conductance** parameterization; if a different option is selected, the data specified in converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the block description for more information on the conversion.

This parameter determines which measures of valve opening you must specify—and therefore which of those measures appear as parameters in the block dialog box.

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions
`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value generally reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum
`0.3` (default) | positive unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

Leakage area — Opening area in the maximally closed position due to sealing imperfections

`1e-12 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Opening area`.

Cross-sectional area at ports A and B — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be the same in size. The flow area specified here should ideally match those of the inlets of adjoining components.

Laminar flow pressure ratio — Pressure ratio at which the flow transitions between laminar and turbulent

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

Reference temperature — Inlet temperature used in the measurement of sonic conductance

`293.15 K` (default) | positive scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance
`1.185` (default) | positive scalar in units of density

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Smoothing factor` — Amount of smoothing to apply to the valve opening area function
`0` (default)

Amount of smoothing to apply to the opening area function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening area function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings), which tend to slow down the rate of simulation.

See Also

Gate Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

Ball Valve (TL)

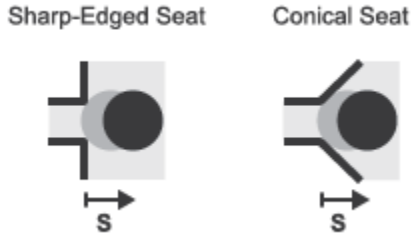
Flow control valve actuated by longitudinal motion of ball element

Library

Thermal Liquid/Valves & Orifices/Flow Control Valves

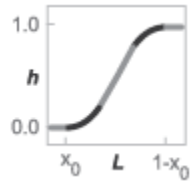
Description

The Ball Valve (TL) block models the flow reduction due to a ball valve in a thermal liquid network. The valve consists of a ball on a perforated seat with a cylindrical or conical shape. The valve opens when the ball undergoes a positive displacement from its seat, allowing fluid to flow through the seat perforation.



Ball Valve Seat Types

A smoothing function allows the valve opening area to change smoothly between the fully closed and fully open positions. The smoothing function does this by removing the curve discontinuities at the zero and maximum ball positions. The figure shows the effect of smoothing on the valve opening area curve.



Opening-Area Curve Smoothing

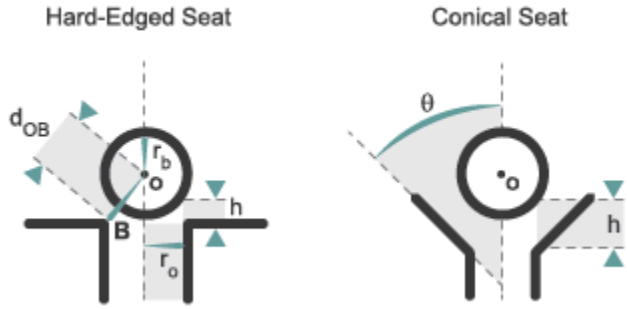
Valve Opening Area

The block computes the valve opening area directly from valve geometry parameters. The calculation depends on the **Valve seat specification** parameter setting. If the valve seat is set to `Sharp-edged`, the valve opening area is based on the geometrical expression:

$$A = \pi r_o \left(1 - \left(\frac{r_b}{d_{OB}} \right)^2 \right) d_{OB}(h),$$

where:

- S is the valve opening area.
- r_o is the valve orifice radius.
- r_b is the valve ball radius.
- $d_{OB}(h)$ is the distance between the center of the ball (point **O** in the figure) and the edge of the orifice (point **B**). This distance is a function of the valve lift (h).



Valve Parameters

If the valve seat is set to `Conical`, the valve opening area is based on the geometrical expression:

$$A = \pi r_b \sin(\theta) h + \frac{\pi}{2} \sin\left(\frac{\theta}{2}\right) \sin(\theta) h^2,$$

where:

- θ is the angle between the conical seat wall and centerline.

The valve opening expressions introduce undesirable discontinuities at the fully open and fully closed positions. The block eliminates these discontinuities using polynomial expressions that smooth the transitions to and from the fully open and fully closed positions. The valve smoothing expressions are

$$\lambda_L = 3\bar{h}_L^2 - 2\bar{h}_L^3$$

and

$$\lambda_R = 3\bar{h}_R^2 - 2\bar{h}_R^3$$

where:

$$\bar{h}_L = \frac{h}{\Delta h_{smooth}}$$

and

$$\bar{h} = \frac{h - \left(h_{max} - \Delta h_{smooth}\right)}{h_{max} - \left(h_{max} - \Delta h_{smooth}\right)}$$

In the equations:

- λ_L is the smoothing expression for the fully closed portion of the valve opening curve.
- λ_R is the smoothing expression applied to the fully open portion of the valve opening curve.
- h_{Max} is the maximum valve lift.
- Δh_{smooth} is the valve lift smoothing region:

  $$\Delta h_{smooth} = f_{smooth} \frac{h_{Max}}{2}$$

  where f_{smooth} is a smoothing factor between 0 and 1.

The smoothed valve opening area is given by the piecewise conditional expression

$$S_R = \begin{cases} S_{Leak}, & h \le 0 \\ S_{Leak}\left(1 - \lambda_L\right) + \left(A + S_{Leak}\right)\lambda_L, & h < \Delta h_{smooth} \\ A + S_{Leak}, & h \le h_{Max} - \Delta h_{smooth}, \\ \left(A + S_{Leak}\right)\left(1 - \lambda_R\right) + \left(S_{Leak} + S_{Max}\right)\lambda_R, & h < h_{Max} \\ S_{Leak} + S_{Max}, & h \ge h_{Max} \end{cases}$$

where:

- S_R is the smoothed valve opening area.
- S_{Leak} is the valve leakage area.
- S_{Max} is the maximum valve opening area:

  $$S_{Max} = \pi r_o^2$$

Mass Balance

The mass conservation equation in the valve is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- $\dot{m}_A$ is the mass flow rate into the valve through port A.
- $\dot{m}_B$ is the mass flow rate into the valve through port B.

Energy Balance

The energy conservation equation in the valve is

$$\phi_A + \phi_B = 0,$$

where:

- ϕ_A is the energy flow rate into the valve through port A.
- ϕ_B is the energy flow rate into the valve through port B.

Momentum Balance

The momentum conservation equation in the valve is

$$p_A - p_B = \frac{\dot{m}\sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{2\rho_{Avg}C_d^2S^2}\left[1 - \left(\frac{S_R}{S}\right)^2\right]PR_{Loss},$$

where:

- p_A and p_B are the pressures at port A and port B.
- $\dot{m}$ is the mass flow rate.
- $\dot{m}_{cr}$ is the critical mass flow rate:

$$\dot{m}_{cr} = \mathrm{Re}_{cr}\,\mu_{Avg}\sqrt{\frac{\pi}{4}S_R}.$$

- ρ_{Avg} is the average liquid density.
- C_{d} is the discharge coefficient.
- S is the valve inlet area.
- PR_{Loss} is the pressure ratio:

$$PR_{Loss} = \frac{\sqrt{1-\left(S_R/S\right)^2\left(1-C_d^2\right)} - C_d\left(S_R/S\right)}{\sqrt{1-\left(S_R/S\right)^2\left(1-C_d^2\right)} + C_d\left(S_R/S\right)}.$$

Parameters

Parameters Tab

Valve seat specification

Choice of valve seat geometry. Options include `Sharp-edged` and `Conical`. The default setting is `Sharp-edged`.

Cone angle

Angle formed by the sides of the conical seat. This parameter is active only when the **Valve seat specification** parameter is active. The default value is `120` deg.

Ball diameter

Diameter of the spherical control member. The default value is `0.01` m.

Orifice diameter

Diameter of the valve opening. The default value is `7e-3` m.

Ball displacement offset

Control member offset from the zero position. The control member displacement is the sum of the input signal S and the displacement offset specified. The default value is `0` m.

Leakage area

Area through which fluid can flow in the fully closed valve position. This area accounts for leakage between the valve inlets. The default value is `1e-12` m^2.

Smoothing factor

Portion of the opening-area curve to smooth expressed as a fraction. Smoothing eliminates discontinuities at the minimum and maximum flow valve positions. The smoothing factor must be between `0` and `1`. Enter a value of `0` for zero smoothing. Enter a value of `1` for full-curve smoothing. The default value is `0.01`.

Cross-sectional area at ports A and B

Area normal to the direction of flow at the valve inlets. This area is assumed the same for all the inlets. The default value is `0.01` m^2.

Characteristic longitudinal length

Distance traversed by the fluid between the valve inlets. The default value is `0.1` m^2.

Discharge coefficient

Ratio of the actual mass flow rate through the valve to its ideal, or theoretical, value. The discharge coefficient accounts for the effects of valve geometry. The value must be between `0` and `1`.

Critical Reynolds number

Reynolds number at which flow transitions between laminar and turbulent regimes. Flow is laminar below this number and turbulent above it. The default value is `12`.

Variables Tab

Mass flow rate into port A

Mass flow rate into the component through port **A** at the start of simulation. The default value is `1 kg/s`.

Ports

- A — Thermal liquid conserving port representing valve inlet A
- B — Thermal liquid conserving port representing valve inlet B
- S — Physical signal input port for the control member displacement

See Also

Gate Valve (TL)

Introduced in R2016a

Cartridge Valve Insert

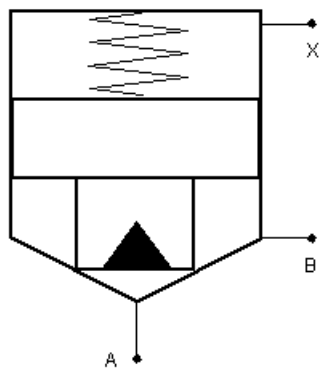
Hydraulic cartridge valve insert

Library

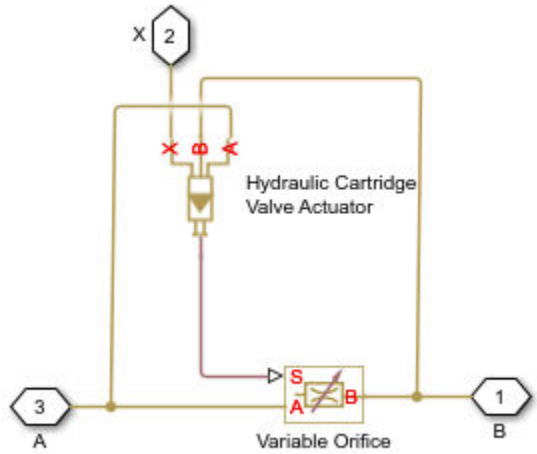
Directional Valves

Description

The Cartridge Valve Insert block represents an insert of a hydraulic cartridge valve consisting of a poppet interacting with the seat. The poppet position is determined by pressures at ports A, B, and X and force of the spring. A schematic diagram of the cartridge valve insert is shown in the following illustration.



The Cartridge Valve Insert block is a structural model consisting of a Hydraulic Cartridge Valve Actuator block and a Variable Orifice block, as shown in the next illustration.



Pressures at port A and port B tend to open the valve, while pressure at the control port X, together with the spring, acts to close it. The model does not account for flow rates caused by poppet displacement and any loading on the poppet, such as inertia and friction. The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke. Hydraulic properties of the gap between the poppet and the seat are simulated with the Variable Orifice block.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve control terminal, respectively. The block positive direction is from port A to port B. Pressure at port X acts to close the valve, while pressures at port A and port B act to open the orifice.

Basic Assumptions and Limitations

- Valve opening is linearly proportional to the pressure differential.
- No loading on the poppet, such as inertia or friction, is considered.
- The model does not account for flow rates caused by poppet displacement.

- For orifices specified by the passage area (the first two parameterization options), the transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- For orifices specified by pressure-flow characteristics (the third parameterization option), the model does not explicitly account for the flow regime or leakage flow rate because the tabulated data is assumed to account for these characteristics.

Parameters

Port A poppet area

Effective poppet area at port A. The parameter value must be greater than zero. The default value is `2e-4` m^2.

Port A to port X area ratio

Ratio between poppet areas at port A and port X. The parameter value must be greater than zero. The default value is `0.66`.

Preload force

Spring preload force. The default value is `26` N.

Spring rate

Spring rate. The default value is `1.4e4` N/m.

Poppet stroke

Maximum poppet stroke. The parameter value must be greater than or equal to zero. The default value is `5e-3` m. This parameter is used if **Orifice specification** is set to `By maximum area and opening`.

Initial opening

The initial opening of the valve. Its value must be greater than or equal to zero. The default value is `0`.

Orifice specification

Select one of the following methods for specifying the hydraulic properties of the gap between the poppet and the seat:

- `By maximum area and opening` — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- `By area vs. opening table` — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of orifice openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

For more information on these options, see the Variable Orifice block reference page.

Orifice maximum area

Specify the area of a fully opened orifice. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Orifice specification** is set to `By maximum area and opening`.

Tabulated orifice openings

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Orifice specification** is set to `By area vs. opening table`, the **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup. If **Orifice specification** is set to `By pressure-flow characteristic`, the **Tabulated orifice openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Tabulated orifice area

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Orifice specification** is set to `By area vs. opening table`.

Tabulated pressure differentials

Specify the pressure differential vector as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default

values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Orifice specification** is set to `By pressure-flow characteristic`.

Tabulated flow rates

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of orifice openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of orifice opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Orifice specification** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Time constant

Sets the time constant of the first-order lag, which is introduced between the required and the actual poppet positions to account for actuator dynamics. The parameter value must be greater than zero. The default value is `0.01` s.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Orifice specification**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Orifice specification** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port associated with the valve control terminal.

See Also

Cartridge Valve Insert with Conical Seat | Check Valve | Hydraulic Cartridge Valve Actuator | Pilot-Operated Check Valve

Introduced in R2008a

Cartridge Valve Insert with Conical Seat

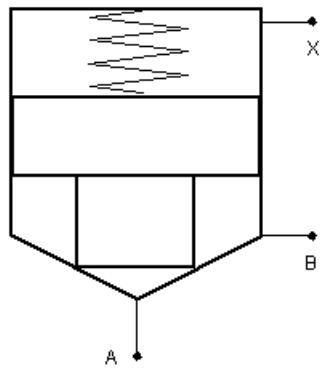
Hydraulic cartridge valve insert with conical seat

Library

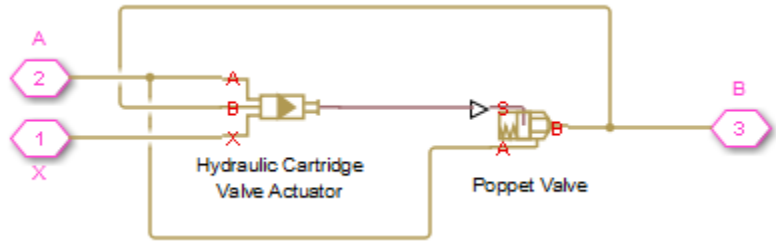
Directional Valves

Description

The Cartridge Valve Insert with Conical Seat block represents an insert of a hydraulic cartridge valve consisting of a poppet interacting with the conical seat. The poppet position is determined by pressures at ports A, B, and X and force of the spring. A schematic diagram of the cartridge valve insert with conical seat is shown in the following illustration.



The Cartridge Valve Insert with Conical Seat block is a structural model consisting of a Hydraulic Cartridge Valve Actuator block and a Poppet Valve block, as shown in the next illustration.



Pressures at port A and port B tend to open the valve, while pressure at the control port X, together with the spring, acts to close it. The model does not account for flow rates caused by poppet displacement and any loading on the poppet, such as inertia and friction. The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke. Hydraulic properties of the gap between the poppet and the seat are simulated with the Poppet Valve block.

Connections A, B, and X are hydraulic conserving ports associated with the valve inlet, valve outlet, and valve control terminal, respectively. The block positive direction is from port A to port B. Pressure at port X acts to close the valve, while pressures at port A and port B act to open the orifice.

Basic Assumptions and Limitations

- Valve opening is linearly proportional to the pressure differential.
- No loading on the poppet, such as inertia or friction, is considered.
- The model does not account for flow consumption caused by poppet displacement.

Parameters

Port A poppet area

Effective poppet area at port A. The parameter value must be greater than zero. The default value is `2e-4` m^2.

Port A to port X area ratio

Ratio between poppet areas at port A and port X. The parameter value must be greater than zero. The default value is `0.66`.

Preload force

Spring preload force. The default value is `26` N.

Spring rate

Spring rate. The default value is `1.4e4` N/m.

Poppet stroke

Maximum poppet stroke. The parameter value must be greater than zero. The default value is `0.005` m.

Poppet diameter

Maximum poppet diameter. The parameter value must be greater than or equal to zero. The default value is `0.01` m.

Seat cone angle

The cone angle of the valve seat. The default value is `120` degrees.

Initial opening

The initial opening of the valve. Its value must be greater than or equal to zero. The default value is `0`.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.

- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Time constant

Sets the time constant of the first-order lag, which is introduced between the required and the actual poppet positions to account for actuator dynamics. The parameter value must be greater than zero. The default value is `0.01` s.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port associated with the valve control terminal.

Examples

For an example of using this block, see the Injection Molding Actuation System example.

See Also

Cartridge Valve Insert | Hydraulic Cartridge Valve Actuator | Poppet Valve

Introduced in R2010a

Centrifugal Force in Rotating Cylinder

Centrifugal force in rotating hydraulic cylinders

Library

Hydraulic Cylinders

Description

The Centrifugal Force in Rotating Cylinder block is a building block to be used in hydraulic cylinder models, to account for centrifugal forces exerted by fluid on a piston, if the cylinder rotates about its symmetry axis. Such cylinders are used in control mechanisms of various friction clutches, brakes, square-jaw positive clutches, dog clutches, and so on. No inertial effects are considered in the model. In other words, the angular velocity is assumed to be constant or changing at very low speed.

The centrifugal force is computed with the following equation:

$$F = \frac{\pi\rho\omega^2}{4}\left(r_o^4 - r_i^4 - 2r_p^2\left(r_o^2 - r_i^2\right)\right)$$

where

F	Centrifugal force
r_o	Piston outer radius
r_i	Piston inner radius
r_p	Fluid entry radius. For design purposes, $r_p < \sqrt{\left(r_o^2 + r_i^2\right)/2}$

ρ	Fluid density
ω	Shaft angular velocity

Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and case, respectively. Connection W is a physical signal port through which shaft angular velocity is imported. The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

Basic Assumptions and Limitations

- No inertial effects are considered in the model; that is, the angular velocity of the cylinder is assumed to be constant or changing at very low speed.
- Fluid inertia is not taken into account.
- Fluid compressibility is not taken into account.

Parameters

Piston outer radius

The default value is `0.1` m.

Piston inner radius

The default value is `0.05` m.

Fluid entry radius

The default value is `0.06` m.

Fluid density

The default value is `880` kg/m^3.

Cylinder orientation

Specifies block orientation with respect to the globally assigned positive direction. The block can be installed in two different ways, depending upon whether it exerts force in the positive or in the negative direction. If positive velocity applied at port W exerts force in negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

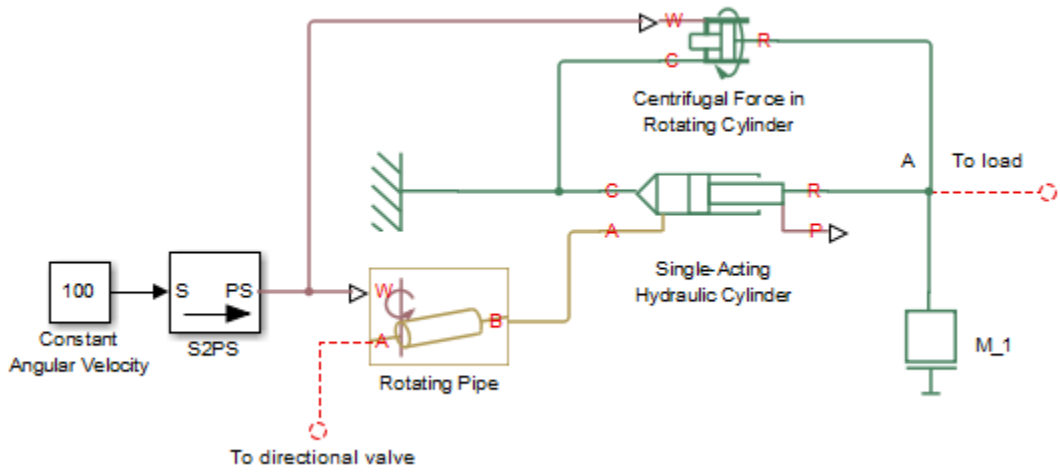
When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Cylinder orientation**

All other block parameters are available for modification.

Examples

The following schematic diagram shows a model of a custom single-acting hydraulic cylinder built of the Single-Acting Hydraulic Cylinder, Rotating Pipe, and Centrifugal Force in Rotating Cylinder blocks.



The centrifugal force generated by the Centrifugal Force in Rotating Cylinder block sums up with that of the cylinder at node A. The Rotating Pipe block simulates the pipeline between the cylinder chamber and the channel in the center of a rotating shaft where the cylinder is installed.

Ports

The block has the following ports:

`R`

Mechanical translational conserving port associated with the cylinder rod.

`C`

Mechanical translational conserving port associated with the cylinder clamping structure.

`W`

Physical signal port providing the angular velocity value.

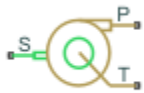
See Also

Rotating Pipe

Introduced in R2010a

Centrifugal Pump

Centrifugal pump with choice of parameterization options



Library

Pumps and Motors

Description

The Centrifugal Pump block represents a centrifugal pump of any type as a data-sheet-based model. Depending on data listed in the manufacturer's catalog or data sheet for your particular pump, you can choose one of the following model parameterization options:

- `By approximating polynomial` — Provide values for the polynomial coefficients. These values can be determined analytically or experimentally, depending on the data available. This is the default method.
- `By two 1D characteristics: P-Q and N-Q` — Provide tabulated data of pressure differential P and brake power N versus pump delivery Q characteristics. The pressure differential and brake power are determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By two 2D characteristics: P-Q-W and N-Q-W` — Provide tabulated data of pressure differential P and brake power N versus pump delivery Q characteristics at different angular velocities W. The pressure differential and brake power are determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

These parameterization options are further described in greater detail:

- “Parameterizing the Pump by Approximating Polynomial” on page 1-337
- “Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery” on page 1-342
- “Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery at Different Angular Velocities” on page 1-343

Connections P and T are hydraulic conserving ports associated with the pump outlet and inlet, respectively. Connection S is a mechanical rotational conserving port associated with the pump driving shaft. The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P as its driving shaft S rotates in the globally assigned positive direction.

Parameterizing the Pump by Approximating Polynomial

If you set the **Model parameterization** parameter to `By approximating polynomial`, the pump is parameterized with the polynomial whose coefficients are determined, analytically or experimentally, for a specific angular velocity depending on the data available. The pump characteristics at other angular velocities are determined using the affinity laws.

The approximating polynomial is derived from the Euler pulse moment equation [1 on page 1-349, 2 on page 1-349], which for a given pump, angular velocity, and fluid can be represented as the following:

$$p_{ref} = k \cdot p_E - p_{HL} - p_D \tag{1-1}$$

where

p_{ref}	Pressure differential across the pump for the reference regime, characterized by the reference angular velocity and density
k	Correction factor. The factor is introduced to account for dimensional fluctuations, blade incongruity, blade volumes, fluid internal friction, and so on. The factor should be set to 1 if the approximating coefficients are determined experimentally.
p_{E}	Euler pressure
p_{HL}	Pressure loss due to hydraulic losses in the pump passages

p_D	Pressure loss caused by deviations of the pump delivery from its nominal (rated) value

The Euler pressure, p_E, is determined with the Euler equation for centrifugal machines [1 on page 1-349, 2 on page 1-349] based on known pump dimensions. For an existing pump, operating at constant angular velocity and specific fluid, the Euler pressure can be approximated with the equation

$$p_E = \rho_{ref} \left(c_0 - c_1 \cdot q_{ref} \right)$$

where

ρ_{ref}	Fluid density
c_0, c_1	Approximating coefficients. They can be determined either analytically from the Euler equation [1 on page 1-349, 2 on page 1-349] or experimentally.
q_{ref}	Pump volumetric delivery at reference regime

The pressure loss due to hydraulic losses in the pump passages, p_{HL}, is approximated with the equation

$$p_{HL} = \rho_{ref} \cdot c_2 \cdot q_{ref}^2$$

where

ρ_{ref}	Fluid density
c_2	Approximating coefficient
q_{ref}	Pump volumetric delivery at reference regime

The blade profile is determined for a specific fluid velocity, and deviation from this velocity results in pressure loss due to inconsistency between the fluid velocity and blade profile velocity. This pressure loss, p_D, is estimated with the equation

$$p_D = \rho_{ref} \cdot c_3 \left(q_D - q_{ref} \right)^2$$

where

ρ_{ref}	Fluid density
c_3	Approximating coefficient
q_{ref}	Pump volumetric delivery at reference regime
q_D	Pump design delivery (nominal delivery)

The resulting approximating polynomial takes the form:

$$p_{ref} = \rho_{ref}\left(k(c_0 - c_1 q_{ref}) - c_2 {q_{ref}}^2 - c_3\left(q_D - q_{ref}\right)^2\right) \tag{1-2}$$

The pump characteristics, approximated with four coefficients c_0, c_1, c_2, and c_3, are determined for a specific fluid and a specific angular velocity of the pump's driving shaft. These two parameters correspond, respectively, to the **Reference density** and **Reference angular velocity** parameters in the block dialog box. To apply the characteristics for another velocity ω or density ρ, the affinity laws are used. With these laws, the delivery at reference regime, which corresponds to given pump delivery and angular velocity, is computed with the expression

$$q_{ref} = q\frac{\omega_{ref}}{\omega} \tag{1-3}$$

where q and ω are the instantaneous values of the pump delivery and angular velocity. Then the pressure differential p_{ref} at reference regime computed with “Equation 1-2” and converted into pressure differential p at current angular velocity and density

$$p = p_{ref} \cdot \left(\frac{\omega}{\omega_{ref}}\right)^2 \cdot \frac{\rho}{\rho_{ref}}$$

“Equation 1-2” describes pump characteristic for ω > 0 and q >= 0. Outside this range, the characteristic is approximated with the following relationships:

$$p = \begin{cases} -k_{leak} \cdot q & \text{for } \omega <= 0 \\ p_{\max} - k_{leak} \cdot q & \text{for } \omega > 0, q < 0 \\ -k_{leak} \cdot \left(q - q_{\max}\right) & \text{for } \omega > 0, q > q_{\max} \end{cases} \tag{1-4}$$

$$q_{\max} = \frac{-b + \sqrt{b^2 + 4ac}}{2a}$$

$$a = \left(c_2 + c_3\right) \cdot \alpha^2$$

$$b = \left(k \cdot c_1 - 2c_3 \cdot q_D\right) \cdot \alpha$$

$$c = k \cdot c_0 - c_3 \cdot q_D^2$$

$$\alpha = \frac{\omega}{\omega_{ref}}$$

$$q_{\max} = \rho \frac{1}{\alpha^2}\left(k \cdot c_0 - c_3 \cdot q_D^2\right)$$

where

k_{leak}	Leakage resistance coefficient
$q_{\max}$	Maximum pump delivery at given angular velocity. The delivery is determined from “Equation 1-2” at $p = 0$.
$p_{\max}$	Maximum pump pressure at given angular velocity. The pressure is determined from “Equation 1-2” at $q = 0$.
k	Correction factor, as described in “Equation 1-1”.

The hydraulic power at the pump outlet at reference conditions is

$$N_{hyd} = p_{ref} \cdot q_{ref}$$

The output hydraulic power at arbitrary angular velocity and density is determined with the affinity laws

$$N = N_{ref}\left(\frac{\omega}{\omega_{ref}}\right) \cdot \frac{\rho}{\rho_{ref}}$$

The power at the pump driving shaft consists of the theoretical hydraulic power (power before losses associated with hydraulic loss and deviation from the design delivery) and friction loss at the driving shaft. The theoretical hydraulic power is approximated using the Euler pressure

$$N_{hyd0} = p_{Eref} \cdot q_{ref} \cdot \left(\frac{\omega}{\omega_{ref}} \right)^3$$

where

N_{hyd0}	Pump theoretical hydraulic power
p_{Eref}	Euler pressure. The theoretical pressure developed by the pump before losses associated with hydraulic loss and deviation from the design delivery.

The friction losses are approximated with the relationship:

$$N_{fr} = \left(T_0 + k_p \cdot p \right) \cdot \omega$$

where

N_{fr}	Friction loss power
T_0	Constant torque at driving shaft associated with shaft bearings, seal friction, and so on
k_p	Torque-pressure relationship, which characterizes the influence of pressure on the driving shaft torque

The power and torque at the pump driving shaft (brake power N_{mech} and brake torque T) are

$$N_{mech} = N_{hyd0} + N_{fr}$$

$$T = \frac{N_{mech}}{\omega}$$

The pump total efficiency η is computed as

$$\eta = \frac{N_{hyd}}{N_{mech}}$$

Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery

If you set the **Model parameterization** parameter to `By two 1D characteristics: P-Q and N-Q`, the pump characteristics are computed by using two one-dimensional table lookups: for the pressure differential based on the pump delivery and for the pump brake power based on the pump delivery. Both characteristics are specified at the same angular velocity ω_{ref} (**Reference angular velocity**) and the same fluid density ρ_{ref} (**Reference density**).

To compute pressure differential at another angular velocity, affinity laws are used, similar to the first parameterization option. First, the new reference delivery q_{ref} is computed with the expression

$$q_{ref} = q\frac{\omega_{ref}}{\omega}$$

where q is the current pump delivery. Then the pressure differential across the pump at current angular velocity ω and density ρ is computed as

$$p = p_{ref} \cdot \left(\frac{\omega}{\omega_{ref}} \right)^2 \cdot \frac{\rho}{\rho_{ref}}$$

where p_{ref} is the pressure differential determined from the P-Q characteristic at pump delivery q_{ref}.

Brake power is determined with the equation

$$N = N_{ref} \cdot \left(\frac{\omega}{\omega_{ref}} \right)^3 \cdot \frac{\rho}{\rho_{ref}}$$

where N_{ref} is the reference brake power obtained from the N-Q characteristic at pump delivery q_{ref}.

The torque at the pump driving shaft is computed with the equation $T = N / \omega$.

Parameterizing the Pump by Pressure Differential and Brake Power Versus Pump Delivery at Different Angular Velocities

If you set the **Model parameterization** parameter to `By two 2D characteristics: P-Q-W and N-Q-W`, the pump characteristics are read out from two two-dimensional table lookups: for the pressure differential based on the pump delivery and angular velocity and for the pump brake power based on the pump delivery and angular velocity.

Both the pressure differential and brake power are scaled if fluid density ρ is different from the reference density ρ_{ref}, at which characteristics have been obtained

$$p = p_{ref} \cdot \frac{\rho}{\rho_{ref}}$$

$$N = N_{ref} \cdot \frac{\rho}{\rho_{ref}}$$

where p_{ref} and N_{ref} are the pressure differential and brake power obtained from the plots.

Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- The pump rotates in positive direction, with speed that is greater or equal to zero.
- The reverse flow through the pump is allowed only at still shaft.

Parameters

Model parameterization

Select one of the following methods for specifying the pump parameters:

- `By approximating polynomial` — Provide values for the polynomial coefficients. These values can be determined analytically or experimentally,

depending on the data available. The relationship between pump characteristics and angular velocity is determined from the affinity laws. This is the default method.

- `By two 1D characteristics: P-Q and N-Q` — Provide tabulated data of pressure differential and brake power versus pump delivery characteristics. The pressure differential and brake power are determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods. The relationship between pump characteristics and angular velocity is determined from the affinity laws.
- `By two 2D characteristics: P-Q-W and N-Q-W` — Provide tabulated data of pressure differential and brake power versus pump delivery characteristics at different angular velocities. The pressure differential and brake power are determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

First approximating coefficient

Approximating coefficient c_0 in the block description preceding. The default value is `326.8` Pa/(kg/m^3). This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Second approximating coefficient

Approximating coefficient c_1 in the block description preceding. The default value is `3.104e4` Pa*s/kg. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Third approximating coefficient

Approximating coefficient c_2 in the block description preceding. This coefficient accounts for hydraulic losses in the pump. The default value is `1.097e7` Pa*s^2/(kg*m^3). This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Fourth approximating coefficient

Approximating coefficient c_3 in the block description preceding. This coefficient accounts for additional hydraulic losses caused by deviation from the nominal delivery. The default value is `2.136e5` Pa*s^2/(kg*m^3). This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Correction factor

The factor, denoted as k in the block description preceding, accounts for dimensional fluctuations, blade incongruity, blade volumes, fluid internal friction, and other factors

that decrease Euler theoretical pressure. The default value is `0.8`. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Pump design delivery

The pump nominal delivery. The blades profile, pump inlet, and pump outlet are shaped for this particular delivery. Deviation from this delivery causes an increase in hydraulic losses. The default value is `130` lpm. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Reference angular velocity

Angular velocity of the driving shaft, at which the pump characteristics are determined. The default value is `1.77e3` rpm. This parameter is used if **Model parameterization** is set to `By approximating polynomial` or `By two 1D characteristics: P-Q and N-Q`.

Reference density

Fluid density at which the pump characteristics are determined. The default value is `920` kg/m^3.

Leak resistance

Leakage resistance coefficient (see “Equation 1-4”). The default value is `1e+8` Pa/(m^3/s). This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Drive shaft torque

The friction torque on the shaft at zero velocity. The default value is `0.1` N*m. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Torque-pressure coefficient

The coefficient that provides relationship between torque and pump pressure. The default value is `1e-6` N*m/Pa. This parameter is used if **Model parameterization** is set to `By approximating polynomial`.

Pump delivery vector for P-Q table

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of pressure differentials to specify the P-Q pump characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in lpm, are `[0 28 90 130 154 182]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics: P-Q and N-Q`.

Pressure differential across pump vector

Specify the vector of pressure differentials across the pump as a one-dimensional array. The vector will be used together with the pump delivery vector to specify the P-Q pump characteristic. The vector must be of the same size as the pump delivery vector for the P-Q table. The default values, in bar, are `[2.6 2.4 2 1.6 1.2 0.8]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics: P-Q and N-Q`.

Pump delivery vector for N-Q table

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of the pump brake power to specify the N-Q pump characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in lpm, are `[0 20 40 60 80 100 120 140 160]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics: P-Q and N-Q`.

Brake power vector for N-Q table

Specify the vector of pump brake power as a one-dimensional array. The vector will be used together with the pump delivery vector to specify the N-Q pump characteristic. The vector must be of the same size as the pump delivery vector for the N-Q table. The default values, in W, are `[220 280 310 360 390 420 480 500 550]`. This parameter is used if **Model parameterization** is set to `By two 1D characteristics: P-Q and N-Q`.

Pump delivery vector for P-Q and W table

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of angular velocities and the pressure differential matrix to specify the pump P-Q-W characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in lpm, are `[0 50 100 150 200 250 300 350]`. This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Angular velocity vector for P-Q and W table

Specify the vector of angular velocities, as a one-dimensional array, to be used for calculating both the pump P-Q-W and N-Q-W characteristics. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values

for linear interpolation, at least three values for smooth interpolation. The default values, in rpm, are `[3.2e+03 3.3e+03 3.4e+03 3.5e+03]`. This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Pressure differential matrix for P-Q and W table

Specify the pressure differentials across pump as an `m`-by-`n` matrix, where `m` is the number of the P-Q-W pump delivery values and `n` is the number of angular velocities. This matrix will define the pump P-Q-W characteristic together with the pump delivery and angular velocity vectors. Each value in the matrix specifies pressure differential for a specific combination of pump delivery and angular velocity. The matrix size must match the dimensions defined by the pump delivery and angular velocity vectors. The default values, in bar, are:

```
[ 8.3 8.8 9.3 9.9 ;
  7.8 8.3 8.8 9.4 ;
  7.2 7.6 8.2 8.7 ;
  6.5 7 7.5 8 ;
  5.6 6.1 6.6 7.1 ;
  4.7 5.2 5.7 6.2 ;
  3.4 4 4.4 4.9 ;
  2.3 2.7 3.4 3.6 ; ]
```

This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Pump delivery vector for N-Q and W table

Specify the vector of pump deliveries, as a one-dimensional array, to be used together with the vector of angular velocities and the brake power matrix to specify the pump N-Q-W characteristic. The vector values must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in lpm, are `[0 50 100 150 200 250 300 350]`. This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Brake power matrix for N-Q and W table

Specify the pump brake power as an `m`-by-`n` matrix, where `m` is the number of the N-Q-W pump delivery values and `n` is the number of angular velocities. This matrix will define the pump N-Q-W characteristic together with the pump delivery and angular velocity vectors. Each value in the matrix specifies brake power for a specific combination of pump delivery and angular velocity. The matrix size must match the

dimensions defined by the pump delivery and angular velocity vectors. The default values, in W, are:

```
[ 1.223e+03 1.341e+03 1.467e+03 1.6e+03 ;
  1.414e+03 1.551e+03 1.696e+03 1.85e+03 ;
  1.636e+03 1.794e+03 1.962e+03 2.14e+03 ;
  1.941e+03 2.129e+03 2.326e+03 2.54e+03 ;
  2.224e+03 2.439e+03 2.66e+03 2.91e+03 ;
  2.453e+03 2.691e+03 2.947e+03 3.21e+03 ;
  2.757e+03 3.024e+03 3.307e+03 3.608e+03 ;
  2.945e+03 3.23e+03 3.533e+03 3.854e+03 ; ]
```

This parameter is used if **Model parameterization** is set to `By two 2D characteristics: P-Q-W and N-Q-W`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve or surface with continuous first-order derivatives.

This parameter is used if **Model parameterization** is set to `By By two 1D characteristics: P-Q and N-Q` or `By two By two 2D characteristics: P-Q-W and N-Q-W`. For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

This parameter is used if **Model parameterization** is set to `By By two 1D characteristics: P-Q and N-Q` or `By two By two 2D characteristics: P-Q-W and N-Q-W`. For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameter determined by the type of working fluid:

- **Fluid density**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

T

Hydraulic conserving port associated with the pump suction, or inlet.

P

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

References

[1] T.G. Hicks, T.W. Edwards, *Pump Application Engineering*, McGraw-Hill, NY, 1971

[2] I.J. Karassic, J.P. Messina, P. Cooper, C.C. Heald, *Pump Handbook*, Third edition, McGraw-Hill, NY, 2001

See Also

Fixed-Displacement Pump | Variable-Displacement Pressure-Compensated Pump | Variable-Displacement Pump

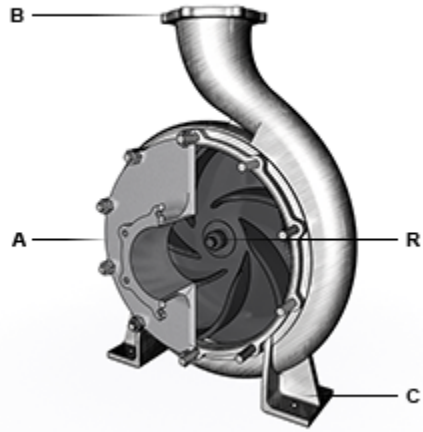
Introduced in R2007a

Centrifugal Pump (TL)

Pressure source based on the centrifugal action of a rotating impeller
Library: Simscape / Fluids / Thermal Liquid / Pumps & Motors

Description

The Centrifugal Pump (TL) block models the pressure rise established across a pump by the centrifugal action of a fanlike rotor, or *impeller*, coupled to a spiral chamber, or *volute*. The pump is parameterized by head and brake power, performance metrics that are often graphed in technical datasheets as functions of flow capacity and impeller shaft speed. Effects attributed to impeller geometry, stage count, and volute type, among other pump design elements, are assumed to be reflected in the performance data. No specific pump architecture or flow direction—axial, radial, or mixed—are assumed.



Side view of a typical centrifugal pump

The pump is powered by an external device—a *prime mover*, often an electrical motor—which spins the impeller shaft (port **R**) against the pump housing (port **C**). The impeller accelerates the flow that it receives from the inlet (port **A**), funneling it through the volute (and, in some cases, a flow diffuser). As it nears the outlet (port **B**), the flow loses speed, causing a pressure rise, as required by the physics of Bernoulli's principle. The flow can momentarily flip direction, causing port **A** to function as the outlet and port **B** as the inlet. However, such conditions are unusual and fall outside of the normal mode of operation.

Mechanical Orientation

The pump generates flow when the impeller shaft is spun in a particular direction, determined by the setting of the **Mechanical orientation** block parameter. If the selected orientation is `Positive`, the impeller shaft rotation (at port **R**) must be positive relative to the pump housing (port **C**). If the selected orientation is `Negative`, the impeller shaft rotation must be negative. No power transmission occurs when the impeller shaft is spun counter to the direction prescribed; the pump is then idle. The positive direction of flow—that generated during normal operation—is always from port **A** to port **B**.

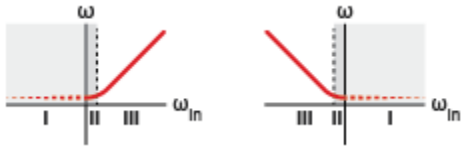
The shaft speed obtained from port **R** relative to port **C** is redefined in the block calculations to be positive whenever its sign matches that prescribed by the **Mechanical orientation** block parameter. Its value is also saturated at a lower threshold bound just slightly above zero. The saturation ensures that the shaft speed cannot flip sign, an event that would allow the pump to transmit power when spun counter to its mechanical orientation. The positive threshold speed ensures that singularities due to division by zero cannot occur and thereby cause simulation to fail. The modified shaft speed is:

$$\omega = \begin{cases} \omega_{\mathrm{Th}}, & \omega_{\mathrm{In}} < 0 \\ (1-\lambda)\,\omega_{\mathrm{Th}} + \lambda\left|\omega_{\mathrm{In}}\right|, & \omega_{\mathrm{In}} < \omega_{\mathrm{Th}} \\ \left|\omega_{\mathrm{In}}\right|, & \omega_{\mathrm{In}} \gtrless \omega_{\mathrm{Th}} \end{cases},$$

where ω is the shaft speed, with the subscript `Th` denoting the threshold value and the subscript `In` denoting the actual, or input, value; ε is the mechanical orientation of the pump, defined as `+1` if positive and `-1` if negative, and λ determines the width of the transition region, which in turn influences the step size taken by the solver during simulation. Generally, the wider the transition region is, the larger the step size can be, and the faster the simulation can progress. The parameter λ is defined as:

$$\lambda = 3\left(\frac{\omega_{\mathrm{In}}}{\omega_{\mathrm{Th}}}\right)^2 - 2\left(\frac{\omega_{\mathrm{In}}}{\omega_{\mathrm{Th}}}\right)^3.$$

The left graph plots the modified shaft speed against the actual value obtained at port **R** relative to port **C** for a pump with positive mechanical orientation. The right graph plots the same speed for a pump with negative mechanical orientation. Region **I** corresponds to a fully saturated shaft speed, region **II** to a partly saturated shaft speed, and region **III** to an unsaturated shaft speed.



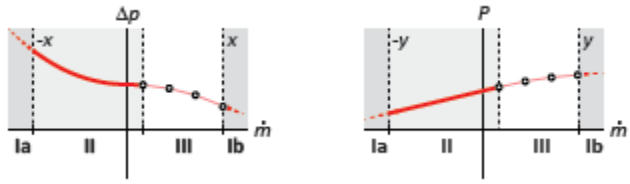
Pump Parameterization

The performance data behind the pump parameterizations is specified in the block in tabulated form. There are two parameterizations: 1-D and 2-D. The 1-D parameterization takes in data on head and brake power, each as a function of flow capacity at some fixed (or *reference*) fluid density and shaft speed. The 2-D parameterization takes in data on the same variables but now as functions also of shaft speed. The choice of parameterization is set by the `Pump parameterization` block parameter.

The dependence of pump performance on shaft speed is captured in both parameterizations via *pump affinity laws*—expressions relating the characteristics of similar pumps moving fluids of different densities and with their impellers running at different speeds. The laws state that flow rate must be proportional to shaft speed, head to the square of shaft speed, and brake power to the cube of shaft speed. They are applied here to a single pump, to convert the head and brake power at the specified reference speed into their proper values at the actual shaft speed.

Head and brake power data are extended in the 1-D parameterization to negative flow capacities. The data extension is based on cubic polynomial regression for head (region **II** of left graph) and on linear regression for brake power (region **II** of right graph). The extension is capped at the negated value of the upper bound on the tabulated flow capacity (-*x* in left graph, -*y* in right graph). No data extension is used in the 2-D parameterization.

Within the tabulated data ranges, head and brake power are determined by linear interpolation of the nearest two breakpoints. Outside of the data ranges (extended, in the 1-D parameterization), they are determined by linear extrapolation of the nearest breakpoint. The extrapolation is limited to positive flow capacity and shaft speed in the 2-D parameterization. Simulation outside of the tabulated data ranges may diminish the accuracy of simulation; where supported, it is intended for the handling of transient dynamics only.



Performance Characteristics

It is common in technical datasheets to characterize pump performance using as variables the head (a length) and flow capacity (a volumetric flow rate). The pump parameterizations provided in the block are, for this reason, based on these variables. Nevertheless, the thermal liquid domain relies on pressure and mass flow rate as its *across* and *through* variables, and the block must therefore convert between the two sets of variables in its calculations.

The pump head given in the datasheets is typically the total *dynamic* head of the pump. Its value is the sum of the static pressure head, the velocity head, and the elevation head. On the scale of the static pressure head of a typical pump, the velocity and elevation heads are generally very small and their values can be rounded off to zero. With this assumption in place, the conversion between the pump head and the pressure rise from inlet to outlet is expressed as:

$$\Delta H = \frac{\Delta p}{\rho g},$$

where ΔH is the total head of the pump, Δp is the static pressure rise across the pump, and ρ and g are the fluid density and gravitational acceleration, respectively. The conversion between flow capacity and mass flow rate is given by:

$$Q = \frac{\dot{m}}{\rho},$$

where Q is the flow capacity and $\dot{m}$ the mass flow rate through the pump.

The flow capacity is determined at the simulated operating conditions from the first pump affinity law:

$$\frac{Q_{\mathrm{R}}}{Q} = \frac{\omega_{\mathrm{R}}}{\omega},$$

where ω is the saturated shaft speed. The subscript R denotes a reference value—either a reference condition reported by the pump manufacturer or a performance variable (here the flow capacity) obtained for those conditions. Expressed in terms of the instantaneous mass flow rate and fluid density, the reference flow capacity becomes:

$$Q_{\mathrm{R}} = \frac{\omega_{\mathrm{R}}}{\omega}\frac{\dot{m}}{\rho},$$

It is this value of the flow capacity that is used during lookup table operations to determine the reference pump head and brake power. In the 2-D parameterization, the instantaneous shaft speed replaces the reference shaft speed in the lookup table operations and the ratio $\omega_{\mathrm{R}}/\omega$ reduces to 1, yielding:

$$Q_{\mathrm{R}} = \frac{\dot{m}}{\rho} (= Q),$$

The pump head is determined at the simulated operating conditions from the second pump affinity law:

$$\frac{\Delta H_{\mathrm{R}}}{\Delta H} = \frac{\omega_{\mathrm{R}}^2}{\omega^2}.$$

In the 1-D parameterization, the reference pump head (ΔH_{R}) is a tabulated function of the reference flow capacity:

$$\Delta H_{\mathrm{R}} = \Delta H(Q_{\mathrm{R}}).$$

In the 2-D parameterization, it is a tabulated function of the instantaneous flow capacity and shaft speed:

$$\Delta H_{\mathrm{R}} = \Delta H(Q,\omega).$$

Expressing ΔH as an equivalent pressure rise and rearranging terms yields:

$$\Delta p = \frac{\omega^2}{\omega_{\mathrm{R}}^2}\rho g \Delta H_{\mathrm{R}}.$$

The shaft speed ratio reduces to 1 in the 2-D parameterization and the calculation becomes:

$$\Delta p = \rho g \Delta H_{\mathrm{R}}.$$

The torque on the impeller shaft is determined at the simulated operating conditions from the third pump affinity law:

$$\frac{P_{\mathrm{R}}}{P} = \frac{\omega_{\mathrm{R}}^3}{\omega^3}\frac{\rho_{\mathrm{R}}}{\rho},$$

where P is brake power. In the 1-D parameterization, the reference brake power (P_{R}) is a tabulated function of the reference flow capacity:

$$P_{\mathrm{R}} = P(Q_{\mathrm{R}}).$$

In the 2-D parameterization, it is a tabulated function of the instantaneous flow capacity and shaft speed:

$$P_{\mathrm{R}} = P(Q,\omega).$$

Brake power and shaft torque are related by the expression:

$$P = T\omega,$$

where T is torque. Expressing the instantaneous brake power in the pump affinity law in terms of the instantaneous shaft torque and rearranging terms yields:

$$T = \frac{\omega^2}{\omega_R^3} \frac{\rho}{\rho_R} P_R .$$

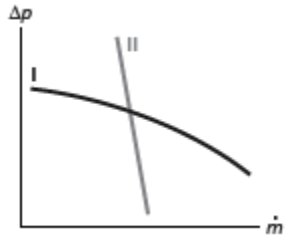
The shaft speed ratio reduces to 1 in the 2-D pump parameterization and the calculation becomes:

$$T = \frac{1}{\omega_R} \frac{\rho}{\rho_R} P_R .$$

Centrifugal and Displacement Pumps

Centrifugal pumps are a type of *rotodynamic* pump. In conjunction with *displacement* pumps, they comprise the majority of pumps currently in use. Rotodynamic pumps work by accelerating the flow in an ``open'' compartment—one never closed off from the ports—and using some of its kinetic energy to generate a pressure rise at the outlet. By contrast, displacement pumps work by trapping a controlled fluid volume in a closed compartment before pushing it through the outlet by the action of a piston, plunger, or other mechanical interface.

The pumps differ in their performance characteristics. The flow rate of a centrifugal pump falls rapidly with a change in head between the ports (curve **I** in the figure). That of a positive-displacement pump varies little (curve **II**). These characteristics lend centrifugal pumps to applications requiring near-constant head and positive-displacement pumps to applications requiring near-constant flow rate. Centrifugal pumps most closely resemble constant pressure sources; positive-displacement pumps most closely resemble constant flow rate sources.



Ports

Conserving

A — Pump inlet
thermal liquid

Opening through which, during normal operation, fluid is sucked into the pump by the centrifugal action of an impeller spinning relative to a mechanical casing.

B — Pump outlet
thermal liquid

Opening through which, during normal operation, fluid is expelled from the pump by the centrifugal action of an impeller spinning relative to a mechanical casing.

C — Pump casing
mechanical rotational

Mechanical enclosure of the pump, used as a local ground against which to measure the rotary action of the impeller shaft.

R — Impeller shaft
mechanical rotational

Rotary part of the of the pump whose motion relative to the pump casing serves to generate flow.

Parameters

`Pump Parameterization` — Method by which to characterize pump performance
`1D tabulated data - head and brake power vs. capacity` (default) | `2D tabulated data - head and brake power vs. capacity and angular velocity`

Method by which to capture the mechanical and flow dynamics of the pump. The methods provided differ in the number of independent of variables that they require (and therefore on the complexity of the data that must be specified). Both parameterizations are based on performance data readily available in pump datasheets. Reference the characteristic curves provided by the pump manufacturer for this data.

`Mechanical orientation` — Correspondence between directions of flow and rotation
`Positive` (default) | `Negative`

Placement of the mechanical portion of the pump relative to the flow portion. The mechanical orientation of the pump determines the direction in which the impeller must rotate in order to generate flow. The required direction is positive if the mechanical orientation is `Positive` also; it is negative if the mechanical orientation is `Negative` instead. Flipping the mechanical orientation of the pump is conceptually equivalent to reversing the spiral direction of the impeller vanes.

`Capacity vector` — Flow capacities at which to specify the pump head and brake power
`[2450, 2800, 3150, 3500] rpm` (default) | vector of *M* positive numbers with units of volume/time

M-element array with the flow capacity breakpoints at which to specify the performance characteristics of the pump. The array values must be greater than or equal to zero and increase monotonically from left to right. When the 1-D pump parameterization is selected, the array must be equal in size to the **Head vector** and **Brake power vector** arrays; when the 2-D pump parameterization is selected, the array must be equal instead to the number of rows in the **Head table** and **Brake power table** matrices.

`Angular speed vector` — Impeller shaft speeds at which to specify the pump head and brake power
`[16.67, 30, 40, 53.33, 66.67, 83.33] lpm` (default) | vector of *N* positive numbers with units of volume/time

N-element array with the impeller shaft speed breakpoints at which to specify the performance characteristics of the pump. The array values must be greater than or equal to zero and increase monotonically from left to right. The array must be equal in size to the number of columns in the **Head table** and **Brake power table** matrices of the 2-D pump parameterization. This parameter is disabled when the 1-D pump parameterization is selected.

`Head vector` — Pressure heads tabulated as functions of flow capacity alone
`[50, 47.5, 45, 40, 35, 25] m` (default) | vector of *M* positive numbers with units of length

M-element array with the pressure heads established from inlet to outlet at the breakpoints defined in the **Capacity vector** array. The pressure heads are specified at a

single impeller shaft speed, which must be obtained from the pump datasheet and entered in the **Reference angular velocity** block parameter. The **Head vector** parameter is disabled when the 2-D pump parameterization is selected.

`Head table` — Pressure heads tabulated as functions of flow capacity and shaft speed

`[25, 32.7, 42.5, 50; 24, 31.7, 41, 49; 22.7, 30.5, 39, 47.5; 20, 28, 36, 45; 15.7, 24, 32, 40; 12.5, 21, 29.5, 37.5] m` (default) | matrix of *M*-by-*N* positive numbers with units of length

M-by-*N* matrix with the pressure heads established from inlet to outlet at the breakpoints defined in the **Capacity vector** and **Angular speed vector** arrays. Each of the *M* matrix rows corresponds to one of the *M* **Capacity vector** array elements. Each of the *N* matrix columns corresponds to one of the *N* **Angular speed vector** array elements. The **Head table** parameter is disabled when the 1-D pump parameterization is selected.

`Brake power vector` — Brake power levels tabulated as functions of flow capacity alone

`[.6, .75, .85, .93, .98, .99] k` (default) | vector of positive numbers with units of power

M-element array with the brake power levels transmitted by the impeller shaft at the breakpoints defined in the **Capacity vector** array. The brake power levels are specified at a single impeller shaft speed, which must be obtained from the pump datasheet and entered in the **Reference angular velocity** block parameter. The **Brake power vector** parameter is disabled when the 2-D pump parameterization is selected.

`Brake power table` — Brake power levels tabulated as functions of flow capacity and shaft speed

`[.28, .38, .5, .6; .31, .41, .55, .68; .33, .44, .6, .75; .36, .48, .65, .83; .38, .53, .72, .93; .39, .55, .75, .97] kW` (default) | matrix of *M*-by-*N* positive numbers with units of power

M-by-*N* matrix with the brake power levels transmitted by the impeller shaft at the breakpoints defined in the **Capacity vector** and **Angular speed vector** arrays. Each of the *M* matrix rows corresponds to one of the *M* **Capacity vector** array elements. Each of the *N* matrix columns corresponds to one of the *N* **Angular speed vector** array elements. The **Head table** parameter is disabled when the 1-D pump parameterization is selected.

`Reference density` — Fluid density for which the pump performance data is reported

`998 kg/m^3` (default) | positive scalar with units of mass/volume

Fluid density at which the tabulated pump performance data is specified. This density is typically appended to the characteristic curves provided by the pump manufacturer. The pump's brake power is scaled during simulation by a factor equal to the ratio of the actual to reference fluid densities (in accordance with the third pump affinity law).

`Reference angular speed` — Impeller shaft speed for which the performance data is reported

`3500 rpm` (default) | positive scalar with units of angle/time

Impeller shaft speed at which the pump performance data is specified. The reference shaft speed is generally appended to the characteristic curves provided by the pump manufacturer. The tabulated head and brake power are scaled during simulation using the second and third pump affinity laws to obtain their instantaneous values at the actual shaft speed.

`Angular speed threshold for flow reversal` — Value at which to saturate the impeller shaft speed

`10 rad/s` (default) | positive scalar with units of angle/time

Minimum speed required of the impeller shaft relative to the impeller casing in order for the pump to generate flow. The shaft speed is saturated at this value as it falls toward zero, with a smooth transition region provided by a cubic polynomial function. The saturation eliminates from the block calculations any singularities caused by division by a zero shaft speed. The smoothing relaxes the solver step size requirements at near-zero shaft speeds, allowing simulation to progress at a faster rate. The width of the transition region (and the impact on the simulation speed) scale with the magnitude of this parameter.

`Cross-sectional area of connecting pipe` — Area normal to the direction of flow at the ports

`0.01 m^2` (default) | positive scalar with units of area

Area normal to the direction of the flow at each of the ports of the pump. The ports are assumed to be identical in size. For best simulation results, the area specified here should match the flow areas of the components adjacent to the pump. Consider using the Sudden Area Change block if you must connect the pump to a component of different flow area.

Variables

`Mass flow rate into port A` — Initial state target for the mass flow rate of the pump

cleared (default) | checked

Desired mass flow rate into the pump through port **A** at the start of simulation. This parameter serves as an initial state target, a guide used by Simscape in assembling the initial configuration of the model. How closely the target is met depends on the constraints imposed by the remainder of the model and on the priority level specified.

See Also

Fixed-Displacement Pump (TL) | Variable-Displacement Pump (TL)

Introduced in R2018a

Check Valve

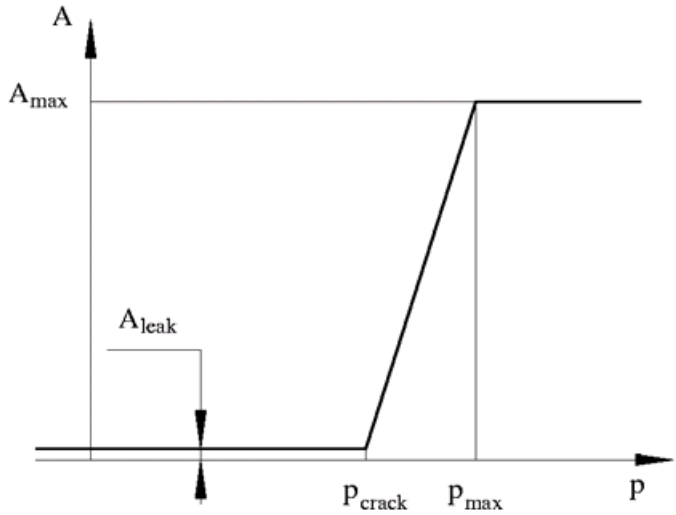
Hydraulic valve that allows flow in one direction only

Library

Directional Valves

Description

The Check Valve block represents a hydraulic check valve as a data-sheet-based model. The purpose of the check valve is to permit flow in one direction and block it in the opposite direction. The following figure shows the typical dependency between the valve passage area A and the pressure differential across the valve $\Delta p = p_A - p_B$, .



The valve remains closed while pressure differential across the valve is lower than the valve cracking pressure. When cracking pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat, thus creating a passage between the inlet and outlet. If the flow rate is high enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the valve passage area is at its maximum. The valve maximum area and the cracking and maximum pressures are generally provided in the catalogs and are the three key parameters of the block.

In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Therefore, the parameter value must be greater than zero.

By default, the block does not include valve opening dynamics, and the valve sets its opening area directly as a function of pressure:

$$A = A(p)$$

Adding valve opening dynamics provides continuous behavior that is more physically realistic, and is particularly helpful in situations with rapid valve opening and closing. The pressure-dependent orifice passage area *A(p)* in the block equations then becomes the steady-state area, and the instantaneous orifice passage area in the flow equation is determined as follows:

$$A(t = 0) = A_{init}$$

$$\frac{dA}{dt} = \frac{A(p) - A}{\tau}$$

In either case, the flow rate through the valve is determined according to the following equations:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

$$A(p) = \begin{cases} A_{leak} & \text{for } p <= p_{crack} \\ A_{leak} + k \cdot (p - p_{crack}) & \text{for } p_{crack} < p < p_{\max} \\ A_{\max} & \text{for } p >= p_{\max} \end{cases}$$

$$k = \frac{A_{\max} - A_{leak}}{p_{\max} - p_{crack}}$$

$$\Delta p = p_A - p_B,$$

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr} \cdot \nu}{C_D \cdot D_H} \right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
A	Instantaneous orifice passage area
$A(p)$	Pressure-dependent orifice passage area
A_{init}	Initial open area of the valve
A_{max}	Fully open valve passage area
A_{leak}	Closed valve leakage area
p_{crack}	Valve cracking pressure
p_{max}	Pressure needed to fully open the valve
p_{cr}	Minimum pressure for turbulent flow
Re_{cr}	Critical Reynolds number
D_H	Instantaneous orifice hydraulic diameter

ρ	Fluid density
ν	Fluid kinematic viscosity
τ	Time constant for the first order response of the valve opening
t	Time

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as

$\Delta p = p_A - p_B$, .

Basic Assumptions and Limitations

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.

Parameters

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is `3e4` Pa.

Maximum opening pressure

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is `1.2e5` Pa.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Opening dynamics

Select one of the following options:

- `Do not include valve opening dynamics` — The valve sets its orifice passage area directly as a function of pressure. If the area changes instantaneously, so does the flow equation. This is the default.
- `Include valve opening dynamics` — Provide continuous behavior that is more physically realistic, by adding a first-order lag during valve opening and closing. Use this option in hydraulic simulations with the local solver for real-time simulation. This option is also helpful if you are interested in valve opening dynamics in variable step simulations.

Opening time constant

The time constant for the first order response of the valve opening. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `0.1` s.

Initial area

The initial opening area of the valve. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `1e-12` m^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Opening dynamics**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the valve inlet.

`B`

Hydraulic conserving port associated with the valve outlet.

Examples

The Hydraulic Flow Rectifier Circuit example illustrates the use of check valves to build a rectifier that keeps the flow passing through a flow control valve always in the same direction, and to select an appropriate orifice depending on the flow direction.

See Also

Pilot-Operated Check Valve

Introduced in R2006a

Check Valve (G)

Valve for limiting flow to a single (forward) direction
Library: Simscape / Fluids / Gas / Valves & Orifices / Directional Control Valves

Description

The Check Valve (G) block models an orifice with a unidirectional opening mechanism to prevent unwanted backflow. The opening mechanism, often spring-loaded by design, responds to pressure, (typically) opening the orifice when the pressure gradient across it falls from inlet (port **A**) to outlet (port **B**), but forcing it shut otherwise. Check valves protect components upstream against pressure surges, temperature spikes, and (in real systems) chemical contamination stemming from points downstream.

The valve opens by degrees, beginning at its cracking pressure, and continuing to the end of its pressure regulation range. The cracking pressure gives the initial resistance, due to friction or spring forces, that the valve must overcome to open by a sliver (or to *crack* open). Below this threshold, the valve is closed and only leakage flow can pass. Past the end of the pressure regulation range, the valve is fully open and the flow at a maximum (determined by the instantaneous pressure conditions).

The cracking pressure assumes an important role in check valves installed upside down. There, the weight of the opening element—such as a ball or piston—and the elevation head of the fluid can act to open the valve. (The elevation head can arise in a model from a pipe upstream of the inlet when it is vertical or given a slant.) A sufficient cracking pressure keeps the valve from opening inadvertently even if placed at a disadvantageous angle.

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Control and Other Pressures

The pressure to which the valve responds is its control pressure. In a typical check valve (and by default in this block), that pressure is the drop from inlet to outlet. This setting ensures that the valve in fact closes if the direction of flow should reverse.

For special cases, an alternative control pressure is provided: the gauge pressure at the inlet. Use it if you know that the inlet will always be at a higher pressure than the outlet (for example, when the inlet connects to a pressure source, such as a pump).

You can select an appropriate control pressure for your model—either **Pressure differential** or `Pressure at port A (gauge)`—using the **Pressure control specification** dropdown list.

When the **Pressure control specification** parameter is set to `Pressure differential`, the control pressure is computed as:

$$p_{\mathrm{Ctl}} = p_{\mathrm{A}} - p_{\mathrm{B}},$$

where p is instantaneous pressure. The subscript `Ctl` denotes the control value and the subscripts **A** and **B** the inlet and outlet, respectively. The port pressures are instantaneous values determined (against absolute zero) during simulation. The cracking pressure is likewise defined as:

$$P_{\mathrm{Crk}} = \left[P_{\mathrm{A}} - P_{\mathrm{B}}\right]_{\mathrm{Crk}},$$

where P is a constant pressure parameter. The subscript `Crk` denotes the cracking value (here a differential). The term in parentheses is obtained as a constant from the **Cracking pressure differential** block parameter. Similarly for the maximum pressure of the valve (at which the valve is fully open):

$$P_{\mathrm{Max}} = \left[P_{\mathrm{A}} - P_{\mathrm{B}}\right]_{\mathrm{Max}},$$

where the subscript `Max` denotes the maximum value of the valve. Here too the term in parentheses is obtained as a constant, from the **Maximum opening pressure differential** block parameter.

When the **Pressure control specification** parameter is set to **Pressure at port A**, the control pressure is computed as:

$$p_{\mathrm{Ctl}} = p_{\mathrm{A}}.$$

The port pressure is an instantaneous value determined (against absolute zero) during simulation. For the cracking pressure:

$$P_{\mathrm{Crk}} = P_{\mathrm{A,Crk}} + P_{\mathrm{Atm}},$$

where the subscript `A,Crk` denotes the cracking value, specified as a gauge pressure at port **A**. This value is obtained as a constant from the **Cracking pressure (gauge)** block parameter. The subscript `Atm` denotes the atmospheric value (specified in the Gas Properties (G) block of the model). The maximum pressure of the valve is:

$$P_{\mathrm{Max}} = P_{\mathrm{A,Max}} + P_{\mathrm{Atm}},$$

where the subscript `A,Max` denotes the maximum value, specified as a gauge pressure at port **A**. This value is obtained as a constant from the **Maximum opening pressure (gauge)** block parameter.

Control Pressure Overshoot

The degree to which the control pressure exceeds the cracking pressure determines how much the valve will open. The pressure overshoot is expressed here as a fraction of the (width of the) pressure regulation range:

$$\hat{p} = \frac{p_{\mathrm{Ctl}} - P_{\mathrm{Crk}}}{P_{\mathrm{Max}} - P_{\mathrm{Crk}}}.$$

The control pressure (p_{Ctl}), cracking pressure (p_{Set}), and maximum opening pressure (P_{Max}) correspond to the control pressure specification chosen (`Pressure differential` or `Pressure at port A`).

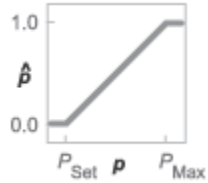
The fraction—technically, the overshoot *normalized*—is valued at `0` in the fully closed valve and `1` in the fully open valve. If the calculation should return a value outside of these bounds, the nearest of the two is used instead. (In other words, the fraction *saturates* at `0` and `1`.)

The normalized control pressure, p, spans three pressure regions. Below the cracking pressure of the valve, its value is a constant zero. Above the maximum pressure of the

same, it is 1. In between, it varies, as a linear function of the (effective) control pressure, p_{Ctl}.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish the simulation run, perhaps excessively so for practical use in real-time simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized pressure overshoot with sharp transitions

The block removes the discontinuities by smoothing them over a specified time scale. The smoothing, which adds a slight distortion to the normalized inlet pressure, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_L = 3\bar{p}_L^2 - 2\bar{p}_L^3$$

and

$$\lambda_R = 3\bar{p}_R^2 - 2\bar{p}_R^3,$$

where

$$\bar{p}_{\mathrm{L}} = \frac{\hat{p}}{\Delta p^{*}}$$

and

$$\bar{p}_{\mathrm{R}} = \frac{\hat{p} - \left(1 - \Delta p^{*}\right)}{\Delta p^{*}}.$$

In the equations:

- $\bar{\lambda}_{\mathrm{L}}$ is the smoothing expression for the transition from the maximally closed position.
- $\bar{\lambda}_{\mathrm{R}}$ is the smoothing expression for the transition from the fully open position.
- Δp^{*} is the (unitless) characteristic width of the pressure smoothing region:

  $$\Delta p^{*} = f^{*}\,\frac{1}{2},$$

  where f^{*} is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

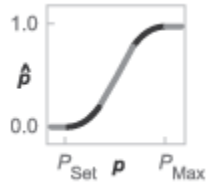
  When the smoothing factor is `0`, the normalized inlet pressure stays in its original form —no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the pressure regulation range (with the normalized inlet pressure taking the shape of an *S*-curve).

  At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the pressure regulation range on each side (for a total smooth region of half the regulation range).

The smoothing adds two new regions to the normalized pressure overshoot—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$\hat{p}^* = \begin{cases} 0, & \hat{p} \le 0 \\ \hat{p}\lambda_L, & \hat{p} < \Delta P^* \\ \hat{p}, & \hat{p} \le 1 - \Delta P^* \\ \hat{p}(1-\lambda_R)+\lambda_R, & \hat{p} < 1 \\ 1 & \hat{p} \ge 1 \end{cases},$$

where the asterisk denotes a smoothed variable (the normalized control pressure overshoot). The figure shows the effect of smoothing on the sharpness of the transitions.



Sonic Conductance

As the normalized control pressure varies during simulation, so does the mass flow rate through the valve. The relationship between the two variables, however, is indirect. The mass flow rate is defined in terms of the valve's sonic conductance and it is this quantity that the normalized inlet pressure determines.

Sonic conductance, if you are unfamiliar with it, describes the ease with which a gas will flow when it is choked—when its velocity is at its theoretical maximum (the local speed of sound). Its measurement and calculation are covered in detail in the ISO 6358 standard (on which this block is based).

Only one value is commonly reported in valve data sheets: one taken at steady state in the fully open position. This is the same specified in the **Sonic conductance at maximum flow** parameter when the **Valve parameterization** setting is `Sonic conductance`. For values across the opening range of the valve, this maximum is scaled by the normalized pressure overshoot:

$$C = (C_{Max} - C_{Min})\,\hat{p} + C_{Min},$$

where C is sonic conductance and the subscripts `Max` and `Min` denote its values in the fully open and fully closed valve.

Because sonic conductance may not be available (or the most convenient choice for your model), the block provides several equivalent parameterizations. Use the **Valve parameterization** drop-down list to select the best for the data at hand. The parameterizations are:

- `Restriction area`
- `Sonic conductance`
- `Cv coefficient (USCS)`
- `Kv coefficient (SI)`

The parameterizations differ only in the data that they require of you. Their mass flow rate calculations are still based on sonic conductance. If you select a parameterization other than `Sonic conductance`, then the block converts the alternate data—the (computed) opening area or a (specified) flow coefficient—into an equivalent sonic conductance.

The flow coefficients measure what is, at bottom, the same quantity—the flow rate through the valve at some agreed-upon temperature and pressure differential. They differ only in the standard conditions used in their definition and in the physical units used in their expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Cv coefficient (USCS)`.
- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `1 bar`; it is expressed in metric units of `m^3/h`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Kv coefficient (SI)`.

If the valve parameterization is set to `Cv Coefficient (USCS)`, the sonic conductance is computed at the maximally closed and fully open valve positions from the **Cv coefficient (SI) at maximum flow** and **Cv coefficient (SI) at leakage flow** block parameters:

$$C = \left(4 \times 10^{-8} C_{\mathrm{v}}\right) \mathrm{m}^3 / (\mathrm{s\ Pa}),$$

where C_{v} is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`. (These are used in the mass flow rate calculations given in the Momentum Balance section.)

If the `Kv coefficient (SI)` parameterization is used instead, the sonic conductance is computed at the same valve positions (maximally closed and fully open) from the **Kv coefficient (USCS) at maximum flow** and **Kv coefficient (USCS) at leakage flow** block parameters:

$$C = \left(4.758 \times 10^{-8} K_{\mathrm{v}}\right) \mathrm{m}^3 / (\mathrm{s\ Pa}),$$

where K_{v} is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`.

For the `Restriction area` parameterization, the sonic conductance is computed (at the same valve positions) from the **Maximum opening area**, and **Leakage area** block parameters:

$$C = \left(0.128 \times 4S / \pi\right) \mathrm{L} / (\mathrm{s\ bar}),$$

where S is the opening area at maximum or leakage flow. The subsonic index, m, is set to `0.5` while the critical pressure ratio, b_{cr} is computed from the expression:

$$0.41 + 0.272 \left[\frac{p\left(S_{\mathrm{Max}} - S_{\mathrm{Leak}}\right) + S_{\mathrm{Leak}}}{S} \right]^{0.25}.$$

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is assumed to reflect entirely in the sonic conductance of the valve (or in the data of the alternate valve parameterizations).

When the flow is choked, the mass flow rate is a function of the sonic conductance of the valve and of the thermodynamic conditions (pressure and temperature) established at the inlet. The function is linear with respect to pressure:

$$\dot{m}_{\mathrm{ch}} = C\rho_0 p_{\mathrm{in}}\sqrt{\frac{T_0}{T_{\mathrm{in}}}},$$

where:

- C is the sonic conductance inside the valve. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- ρ is the gas density, here at standard conditions (subscript `0`), obtained from the **Reference density** block parameter.
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (`in`) or at standard conditions (`0`), the latter obtained from the **Reference temperature** block parameter.

When the flow is subsonic, and therefore no longer choked, the mass flow rate becomes a nonlinear function of pressure—both that at the inlet as well as the reduced value at the outlet. In the turbulent flow regime (with the outlet pressure contained in the back-pressure ratio of the valve), the mass flow rate expression is:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}}\sqrt{\frac{T_0}{T_{\mathrm{in}}}}\left[1-\left(\frac{p_{\mathrm{r}}-b_{\mathrm{cr}}}{1-b_{\mathrm{cr}}}\right)^2\right]^m,$$

where:

- p_{r} is the back-pressure ratio, or that between the outlet pressure (p_{out}) and the inlet pressure (p_{in}):

  $$P_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the flow becomes choked. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

- *m* is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

When the flow is laminar (and still subsonic), the mass flow rate expression changes to:

$$\dot{m}_{\mathrm{lam}} = C\rho_0 p_{\mathrm{in}} \left[\frac{1-p_{\mathrm{r}}}{1-b_{\mathrm{lam}}}\right] \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1-\left(\frac{b_{\mathrm{lam}}-b_{\mathrm{cr}}}{1-b_{\mathrm{cr}}}\right)^2\right]^m$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes (obtained from the **Laminar flow pressure ratio** block parameter). Combining the mass flow rate expressions into a single (piecewise) function, gives:

$$\dot{m} = \begin{cases} \dot{m}_{\mathrm{lam}}, & b_{\mathrm{lam}} \le p_{\mathrm{r}} < 1 \\ \dot{m}_{\mathrm{tur}}, & b_{\mathrm{cr}} \le p_{\mathrm{r}} < p_{\mathrm{lam}} \\ \dot{m}_{\mathrm{ch}}, & p_{\mathrm{r}} < b_{\mathrm{Cr}} \end{cases},$$

with the top row corresponding to subsonic and laminar flow, the middle row to subsonic and turbulent flow, and the bottom row to choked (and therefore sonic) flow.

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of gas can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**. Note that in this block the flow can reach but not exceed sonic speeds.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the gas and the wall that surrounds it. No work is done on or by the gas as it traverses

from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_A + \phi_B = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Valve entrance

gas

Opening through which the working fluid must enter the valve.

B — Valve entrance

gas

Opening through which the working fluid must exit the valve.

Parameters

Pressure control specification — Choice of pressure measurement to use as valve control signal

`Pressure differential` (default) | `Pressure at port A`

Choice of pressure measurement to use as valve control signal. The block uses this setting to determine when the valve should begin to open. In the default setting (`Pressure differential`), the opening pressure of the valve is expressed as a pressure drop from inlet to outlet. In the alternative setting (`Pressure at port A`), it is expressed as an absolute inlet pressure.

Cracking pressure differential — Pressure drop, from inlet to outlet, required to open the valve

`0.01 MPa` (default) | positive scalar in units of pressure

Minimum pressure drop from inlet to outlet required to open the valve. This value marks the beginning of the pressure differential range of the valve (over which it progressively opens to allow for increased flow).

Dependencies

This parameter is active and exposed in the block dialog box when the **Pressure control specification** parameter is set to `Pressure differential`.

`Maximum opening pressure differential` — Pressure drop, from inlet to outlet, at which the valve is fully open

`0.02 MPa` (default) | positive scalar in units of pressure

Pressure drop from inlet to outlet at which the valve is fully open. This value marks the end of the pressure differential range of the valve (over which the same progressively opens to allow for increased flow).

Dependencies

This parameter is active and exposed in the block dialog box when the **Pressure control specification** parameter is set to `Pressure differential`.

`Cracking pressure (gauge)` — Minimum gauge pressure at the inlet required to open the valve

`0.01 MPa` (default) | positive scalar in units of pressure

Minimum gauge pressure at the inlet (port **A**) required to open the valve. This value marks the beginning of the pressure range of the valve (over which the same progressively opens to allow for increased flow).

Dependencies

This parameter is active and exposed in the block dialog box when the **Pressure control specification** parameter is set to `Pressure at port A`.

`Maximum opening pressure (gauge)` — Pressure drop, from inlet to outlet, at which the valve is fully open

`0.02 MPa` (default) | positive scalar in units of pressure

Pressure drop from inlet to outlet at which the valve is fully open. This value marks the end of the pressure differential range of the valve (over which the same progressively opens to allow for increased flow).

Dependencies

This parameter is active and exposed in the block dialog box when the **Pressure control specification** parameter is set to `Pressure at port A`.

`Valve parameterization` — Method by which to characterize the opening of the valve

`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the **Sonic conductance** parameterization; if a different option is selected, the data specified in converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the block description for more information on the conversion.

This parameter determines which measures of valve opening you must specify—and therefore which of those measures appear as parameters in the block dialog box.

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions

`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value generally reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions

`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum

when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Maximum opening area` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position, when the valve is at the upper limit of the pressure regulation range. The block uses this parameter to scale the chosen measure of valve opening—sonic conductance, say, or C_V flow coefficient—throughout the pressure regulation range.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Leakage area` — Opening area in the maximally closed position due to sealing imperfections

`1e-12 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the

valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Cross-sectional area at ports A and B` — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be the same in size. The flow area specified here should ideally match those of the inlets of adjoining components.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance

`293.15 K` (default) | positive scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance

`1.185` (default) | positive scalar in units of density

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Smoothing factor` — Amount of smoothing to apply to the valve opening area function
`0` (default)

Amount of smoothing to apply to the opening area function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening area function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings), which tend to slow down the rate of simulation.

See Also

Pilot-Operated Check Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

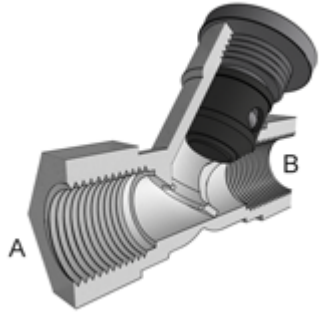
Check Valve (TL)

Valve for the prevention of flow aimed counter to its intended direction

Library: Simscape / Fluids / Thermal Liquid / Valves & Orifices / Directional Control Valves

Description

The Check Valve (TL) block models the opening characteristics of a proportional valve that is forced shut when the pressure difference between its ports is reversed (or, more precisely, dropped below a threshold known as the *cracking pressure*). Check valves are common in backflow prevention devices, such as those used in public water supply networks, where contaminated water downstream of a water main must not be allowed to return upstream. No specific valve shutoff mechanism—whether ball, disc, diaphragm, or other—is assumed in the block. The allowed direction of flow is always from port **A** to port **B**.



Y-Shaped Check Valve with Piston-Type Control Member Partially Retracted

The valve cracks open when the pressure drop across it rises above the cracking pressure specified in the block. The opening area increases linearly with pressure, save for two small pressure intervals near the fully open and fully closed positions (over which nonlinear smoothing is applied in order to remove numerical discontinuities). When the

pressure drop reaches the maximum value specified in the block, the valve is fully open and its opening area no longer increases with pressure. The flow rate through the valve is never truly zero as a small leakage area remains when the pressure falls below the cracking pressure.

Valve Opening Area

The (smoothed) valve opening area is first computed as a linear function of pressure, either that at the valve entrance (port **A**) or the drop between those at the entrance and at the outlet (port **B**). Which of these pressures—termed control pressures—features in the area calculations depends on the setting of the **Pressure specification method** block parameter:

$$p_{\mathrm{Control}} = \begin{cases} p_{\mathrm{A}}, & \text{Pressure at port A method} \\ p_{\mathrm{A}} - p_{\mathrm{B}}, & \text{Pressure differential method} \end{cases},$$

where p is pressure; the subscript `Control` indicates the value to be used in determining the opening area of the valve. Subscripts `A` and `B` denote the thermal liquid ports at which the pressures are obtained. The pressures at the ports are always defined as absolute pressures. The cracking pressure, at which the opening area is at a minimum, is similarly defined:

$$p_{\mathrm{Crack}} = \begin{cases} p_{\mathrm{Crack,G}} + p_{\mathrm{Atm}}, & \text{Pressure at port A method} \\ \Delta p_{\mathrm{Crack}}, & \text{Pressure differential method} \end{cases},$$

where the subscript `Crack` indicates a valve just cracking open, subscript `G` a gauge value, and subscript `Atm` the standard atmospheric value. The value of $p_{\mathrm{Crack,G}}$ is obtained from the **Cracking pressure (gauge)** block parameter; that of $\Delta p_{\mathrm{Crack}}$ is obtained from the **Cracking pressure differential** block parameter. The maximum pressure, at which the opening area is at its largest, is:

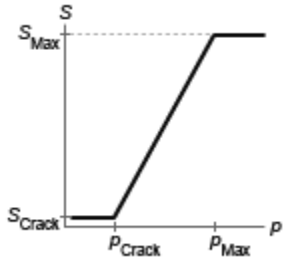
$$p_{\mathrm{Max}} = \begin{cases} p_{\mathrm{Max,G}} + p_{\mathrm{Atm}}, & \text{Pressure at port A method} \\ \Delta p_{\mathrm{Max}}, & \text{Pressure differential method} \end{cases},$$

where subscript `Max` denotes a maximum pressure (obtained as gauge). The value of $p_{\mathrm{Max,G}}$ is obtained from the **Maximum opening pressure (gauge)** block parameter; that of Δp_{Max} is obtained from the **Maximum opening pressure differential** block

parameter. The control, cracking, and maximum pressures give for the linear form of the valve opening area:

$$S_{\mathrm{Lin}} = \frac{S_{\mathrm{Max}} - S_{\mathrm{Crack}}}{p_{\mathrm{Max}} - p_{\mathrm{Crack}}}\left(p_{\mathrm{Ctl}} - p_{\mathrm{Crack}}\right) + S_{\mathrm{Crack}},$$

where *S* denotes the (linear) opening area. The opening area at cracking is equal to the small value specified in the **Leakage area** block parameter. The primary purpose of this parameter is to ensure the numerical robustness of the model by ensuring that no portion of a thermal liquid network becomes completely isolated during simulation.



Valve opening area as a linear function of pressure

The primary purpose of the leakage area of a closed valve is to ensure that at no time does a portion of the hydraulic network become isolated from the remainder of the model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage is generally present in minuscule amounts in real valves but in a model its exact value is less important than it being a small number greater than zero. The leakage area is obtained from the block parameter of the same name.

Opening Area Smoothing

To ensure adequate simulation performance, the valve opening area is smoothed over two small pressure intervals near the specified cracking and maximum pressures. The smoothing is accomplished by means of the polynomial expressions (to be incorporated into the final form of the opening area expression):

$$\lambda_{\text{Crack}} = 3\gamma_{\text{Crack}}^2 - 2\gamma_{\text{Crack}}^3 \qquad \text{and} \qquad \lambda_{\text{Max}} = 3\gamma_{\text{Max}}^2 - 2\gamma_{\text{Max}}^3,$$

where λ is the smoothing factor applied at the cracking (subscript `Crack`) and maximum (subscript `Max`) portions of the surface area expression. The smoothing factor is calculated from the normalized pressure differences γ:

$$\gamma_{\text{Crack}} = \frac{p_{\text{Control}} - p_{\text{Crack}}}{\Delta p_{\text{Smooth}}} \qquad \text{and} \qquad p_{\text{Max}} = \frac{p_{\text{Crack}} - \left(p_{\text{Max}} - \Delta p_{\text{Smooth}}\right)}{\Delta p_{\text{Smooth}}},$$

where Δp_{Smooth} is the pressure smoothing region:

$$\Delta p_{\text{Smooth}} = f_{\text{Smooth}} \frac{p_{\text{Max}} - p_{\text{Crack}}}{2}.$$

The parameter f_{Smooth} is a value between `0` and `1` obtained from the **Smoothing factor** block parameter. The final, smoothed, valve opening area is given by the conditional expression:

$$S_{\text{Smooth}} = \begin{cases} S_{\text{Crack}}, & \text{if } p_{\text{Control}} \le p_{\text{Crack}} \\ S_{\text{Crack}}\left(1 - \lambda_{\text{Crack}}\right) + S\lambda_{\text{Crack}}, & \text{if } p_{\text{Control}} \le p_{\text{Crack}} + \Delta p_{\text{Smooth}} \\ S_{\text{Lin}}, & \text{if } p_{\text{Control}} \le p_{\text{Max}} - \Delta p_{\text{Smooth}} \\ S_{\text{Lin}}\left(1 - \lambda_{\text{Max}}\right) + S_{\text{Max}}\lambda_{\text{Max}}, & \text{if } p_{\text{Control}} \le p_{\text{Max}} \\ S_{\text{Max}}, & \text{if } p_{\text{Control}} \ge p_{\text{Max}} \end{cases}.$$

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of fluid can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through one of the ports (**A** or **B**).

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is captured in the block by the discharge coefficient, a measure of the mass flow rate through the valve relative to the theoretical value that it would have in an ideal valve. Expressing the momentum balance in the valve in terms of the pressure drop induced in the flow:

$$p_A - p_B = \frac{\dot{m}_{Avg}\sqrt{\dot{m}_{Avg}^2 + \dot{m}_{Crit}^2}}{2\rho_{Avg}C_D S_{Smooth}^2}\left[1 - \left(\frac{S_{Smooth}}{S_{Lin}}\right)^2\right]\xi_p,$$

where C_D is the discharge coefficient, and ξ_p is the pressure drop ratio—a measure of the extent to which the pressure recovery at the outlet contributes to the total pressure drop of the valve. The subscript Avg denotes an average of the values at the thermal liquid ports. The critical mass flow rate $\dot{m}_{Crit}$ is calculated from the critical Reynolds number—that at which the flow in the orifice is assumed to transition from laminar to turbulent:

$$\dot{m}_{Crit} = Re_{Crit}\mu_{Avg}\sqrt{\frac{\pi}{4}S_{Lin}},$$

where μ denotes dynamic viscosity. The pressure drop ratio is calculated as:

$$\xi_p = \frac{\sqrt{1 - \left(\frac{S_{Smooth}}{S_{Lin}}\right)^2\left(1 - C_D^2\right)} - C_D\frac{S_{Smooth}}{S_{Lin}}}{\sqrt{1 - \left(\frac{S_{Smooth}}{S_{Lin}}\right)^2\left(1 - C_D^2\right)} + C_D\frac{S_{Smooth}}{S_{Lin}}}.$$

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the fluid and the wall of the valve. No work is done on or by the fluid as it traverses the valve. With these assumptions, energy can enter and exit the valve by advection only, through ports **A** and **B**. By the principle of conservation of energy then, the sum of the energy flows through the ports must always equal zero:

$$\phi_A + \phi_B = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Valve inlet
thermal liquid

Opening through which the flow can enter the valve.

B — Valve outlet
thermal liquid

Opening through which the flow can exit the valve.

Parameters

Pressure control specification — Choice of pressure to use in the block calculations
`Pressure at port A` (default) | `Pressure differential`

Choice of pressure to use in the block calculations. The default setting corresponds to the gauge pressure at the valve inlet (port **A**). The alternative setting corresponds to the pressure differential from inlet (**A**) to outlet (**B**).

Cracking pressure (gauge) — Gauge pressure at which the valve begins to open
`0.1 MPa` (default) | positive scalar in units of pressure

Gauge pressure at port **A** at which the valve begins to open. The valve opening area is saturated at the value of the **Leakage area** block parameter when the pressure at port **A** is at or below the value specified here.

Dependencies

This parameter is active when the **Pressure control specification** block parameter is set to `Pressure at port A`.

`Maximum opening pressure (gauge)` — Gauge pressure at which the valve is fully open

`0.2 MPa` (default) | positive scalar in units of pressure

Gauge pressure at port **A** at which the valve is fully open. The valve opening area is saturated at the value specified in the **Maximum opening area** block parameter when the pressure at port **A** is at or above the value specified here.

Dependencies

This parameter is active when the **Pressure control specification** block parameter is set to `Pressure at port A`.

`Cracking pressure differential` — Pressure differential from port A to port B at which the valve begins to open

`0.1 MPa` (default) | positive scalar in units of pressure

Pressure differential from port A to port B at which the valve begins to open. The valve opening area is saturated at the value of the **Leakage area** block parameter when the pressure differential is at or below the value specified here.

Dependencies

This parameter is active when the **Pressure control specification** block parameter is set to `Pressure differential`.

`Maximum opening pressure differential` — Pressure differential from port A to port B at which the valve is fully open

`0.2 MPa` (default) | positive scalar in units of pressure

Pressure differential from port A to port B at which the valve is fully open. The valve opening area is saturated at the value specified in the **Maximum opening area** block parameter when the pressure differential is at or above the value specified here.

Dependencies

This parameter is active when the **Pressure control specification** block parameter is set to `Pressure differential`.

`Maximum opening area` — Opening area of the valve in the fully open position
`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position. The valve is fully open if the pressure drop from port **A** to port **B** is equal to or greater than that given by the **Maximum opening pressure** block parameter.

`Leakage area` — Opening area of the valve in the maximally closed position
`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

`Smoothing factor` — Measure of the amount of smoothing to apply to the opening area function
`0.01` (default) | positive unitless scalar

Measure of the amount of smoothing to apply to the opening area function. This parameter determines the widths of the regions to be smoothed, one being at the fully open position, the other at the fully closed position. The smoothing superposes on the linear opening area function two nonlinear segments, one for each region of smoothing. The greater the value specified, the greater the smoothing and the broader the nonlinear segments.

`Cross-sectional area at ports A and B` — Flow area at the thermal liquid ports
`0.01 m` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the orifice connects.

`Characteristic longitudinal length` — Measure of the length of the flow path through the valve
`0.1` (default) | positive scalar in units of length

Average distance traversed by the fluid as it moves from inlet to outlet. This distance is used in the calculation of the internal thermal conduction that occurs between the two ports (as part of the smoothed upwind energy scheme employed in the thermal liquid domain).

`Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates
`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes
12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

Variables

`Mass flow rate into port A` — Initial state target for the mass flow rate of the valve
cleared (default) | checked

Desired mass flow rate into the valve through port **A** at the start of simulation. This parameter serves as an initial state target, a guide used by Simscape in assembling the initial configuration of the model. How closely the target is met depends on the constraints imposed by the remainder of the model and on the priority level specified.

See Also

2-Way Directional Valve (TL) | 3-Way Directional Valve (TL) | 4-Way Directional Valve (TL) | Variable Area Orifice (TL)

Introduced in R2016a

Constant Head Tank

Hydraulic reservoir where pressurization and fluid level remain constant regardless of volume change

Library

Low-Pressure Blocks

Description

The Constant Head Tank block represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The size of the tank is assumed to be large enough to neglect the pressurization and fluid level change due to fluid volume. The block accounts for the fluid level elevation with respect to the tank bottom, as well as for pressure loss in the connecting pipe that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

The fluid volume value does not affect the results of simulation. It is introduced merely for information purposes. It is possible for the fluid volume to become negative during simulation, which signals that the fluid volume is not enough for the proper operation of the system. By viewing the results of the simulation, you can determine the extent of the fluid shortage.

For reasons of computational robustness, the pressure loss in the connecting pipe is computed with the equations similar to that used in the Fixed Orifice block:

$$q = \sqrt{\frac{1}{K}} \cdot A_p \sqrt{\frac{2}{\rho}} \cdot \frac{p_{loss}}{\left(p_{loss}^2 + p_{cr}^2\right)^{1/4}}$$

$$p_{cr} = K\frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{d}\right)^2$$

The Critical Reynolds number is set to 15.

The pressure at the tank inlet is computed with the following equations:

$$p = p_{elev} - p_{loss} + p_{pr}$$

$$p_{elev} = \rho \bullet g \bullet H$$

$$A_p = \frac{\pi \bullet d^2}{4}$$

where

p	Pressure at the tank inlet
p_{elev}	Pressure due to fluid level
p_{loss}	Pressure loss in the connecting pipe
p_{pr}	Pressurization
ρ	Fluid density
g	Acceleration of gravity
H	Fluid level with respect to the bottom of the tank
K	Pressure loss coefficient
A_p	Connecting pipe area
d	Connecting pipe diameter
q	Flow rate
p_{cr}	Minimum pressure for turbulent flow

Connection T is a hydraulic conserving port associated with the tank inlet. Connection V is a physical signal port. The flow rate is considered positive if it flows into the tank.

Parameters

Pressurization

Gage pressure acting on the surface of the fluid in the tank. It can be created by a gas cushion, membrane, bladder, or piston, as in bootstrap reservoirs. This parameter must be greater than or equal to zero. The default value is 0, which corresponds to a tank connected to atmosphere.

Fluid level

The fluid level with respect to the tank bottom. This parameter must be greater than zero. The default value is `1` m.

Initial fluid volume

The initial volume of fluid in the tank. This parameter must be greater than zero. The default value is `0.2` m^3.

Inlet pipeline diameter

The diameter of the connecting pipe. This parameter must be greater than zero. The default value is `0.02` m.

Pipeline pressure loss coefficient

The value of the pressure loss coefficient, to account for pressure loss in the connecting pipe. This parameter must be greater than zero. The default value is `1.2`.

Ports

The block has the following ports:

`T`

Hydraulic conserving port associated with the tank inlet.

`V`

Physical signal port that outputs the volume of fluid in the tank.

See Also

Reservoir | Tank

Introduced in R2009a

Counterbalance Valve

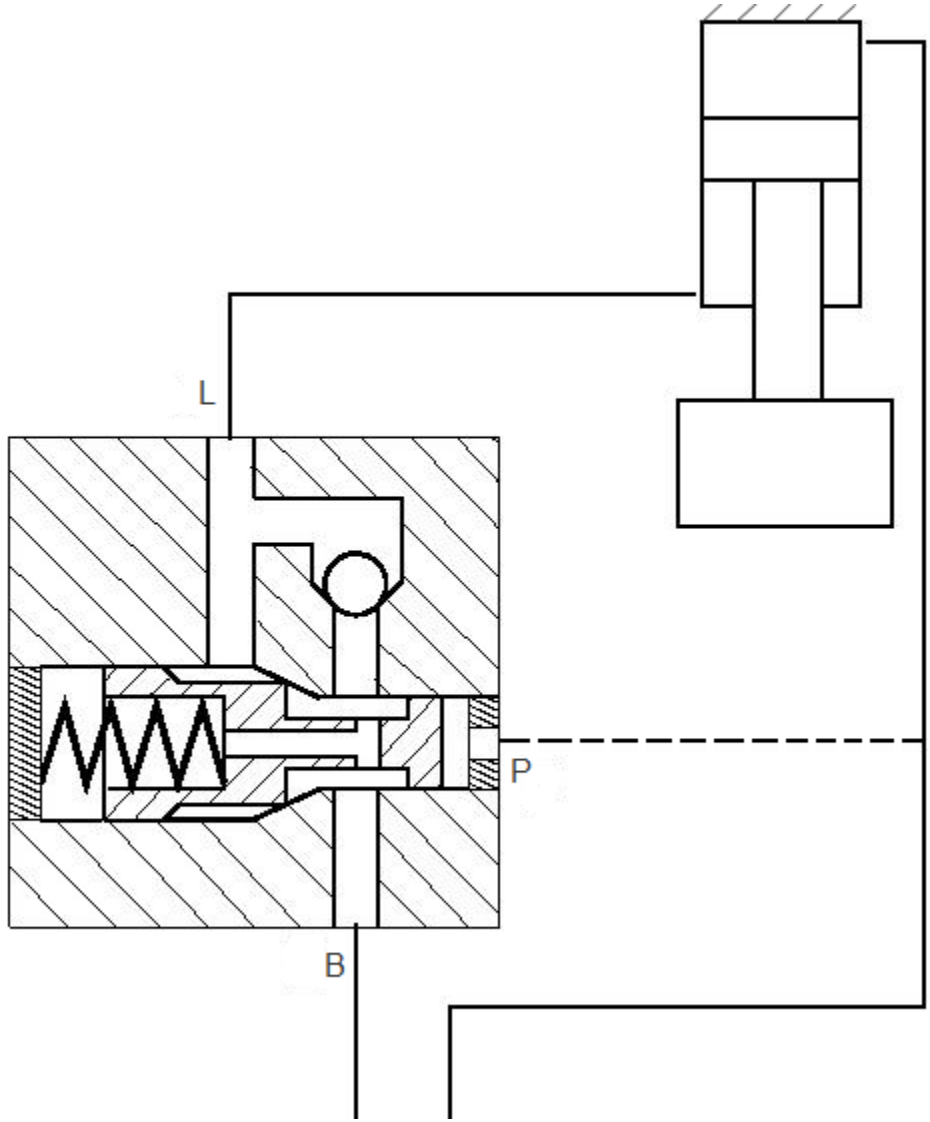
Hydraulic counterbalance valve

Library

Flow Control Valves

Description

Counterbalance valves are used in hydraulic systems working with overriding (running-away) or suspended load. They are designed to create backpressure at the return line of the actuator to prevent losing control over the load. The following illustration shows a counterbalance valve schematic.



If a directional valve (not shown in the picture) is shifted into position that lowers the load, then the fluid from the rod chamber of the cylinder can exit only if pressure at port P (pilot pressure) and port L (load pressure) create enough force to overcome the spring force and open the valve. In statics, the valve is described with the equation

$$F_0 + c \cdot x = p_{pilot} \cdot A_{pilot} + p_{load} \cdot A_{load} - p_{back} \cdot A_{back} \tag{1-5}$$

where

F_0	Spring setting
c	Spring rate
x	Valve opening
p_{pilot}	Pilot pressure (pressure at port P)
p_{load}	Load pressure (pressure at port L)
p_{back}	Backpressure (pressure at return port B)
A_{pilot}	Valve effective area at pilot port P
A_{load}	Valve effective area at load port L
A_{back}	Valve effective area at return port B

Counterbalance valve, classified by type, is an internally-externally piloted valve because both the pilot pressure and the load pressure tend to open the valve. After minor rearrangements, “Equation 1-5” takes the form

$$p_{set} + c_p \cdot x = p_{pilot} \cdot k_{pilot} + p_{load} - p_{back} \cdot k_{back} \tag{1-6}$$

$$p_{set} = F_0 / A_{load}$$
$$c_p = c / A_{load}$$
$$k_{pilot} = A_{pilot} / A_{load}$$
$$k_{back} = A_{back} / A_{load}$$

where

p_{set}	Valve pressure setting
c_{p}	Spring pressure stiffness (Pa/m)
x	Valve opening
k_{pilot}	Pilot ratio
k_{back}	Backpressure ratio

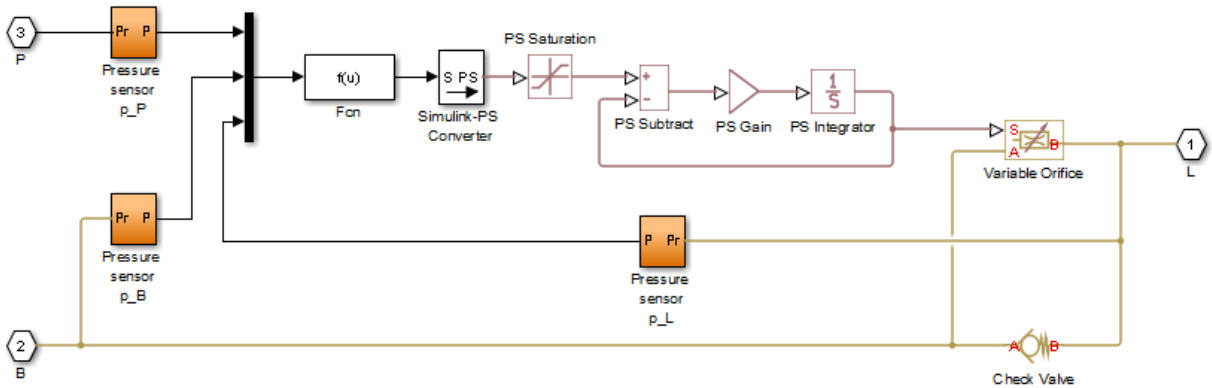
The valve displacement is determined from “Equation 1-6”

$$x = \frac{p_{pilot} \cdot k_{pilot} + p_{load} - p_{back} \cdot k_{back} - p_{set}}{c_p} \qquad (1\text{-}7)$$

$$0 \le x \le x_{max}$$

where x_{max} is the maximum valve displacement.

The Counterbalance Valve block can be represented as the following structural model.



The pressure sensors measure pressure at respective ports and convey their values to the Fcn block, which, together with the PS Saturation block, performs calculations in accordance with “Equation 1-7”. The valve displacement is passed through the first order lag block, built of the PS Subtract, PS Gain, and PS Integrator blocks, to account for valve dynamics. The gain of the PS Gain block is set to 1/*T*, where *T* is the time constant. The Variable Orifice and Check Valve blocks simulate the counterbalance valve orifice and check valve. In the actual Counterbalance Valve block model, the operations performed by the sensors and the Fcn block are executed in the block equation section.

The Counterbalance Valve block is essentially a data-sheet-based model. Depending on data listed in the manufacturer's catalogs or data sheets for your particular valve, you can choose one of the following model parameterization options:

- `By maximum area and opening` — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.

- `By area vs. opening table` — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement.

In the latter case, the PS Saturation block in the structural model is replaced with the PS Lookup Table (1D) block, and you can choose from three interpolation and two extrapolation methods.

Connections L and B are hydraulic conserving ports associated with the load and backpressure ports of the valve. The hydraulic conserving port P is associated with the pilot port. The block positive direction is from port L to port B. Positive pressure at port P opens the valve.

Assumptions and Limitations

- Valve dynamics are approximated by introducing the first order lag between the pressure sensors and the variable orifice control member displacement.
- Inertia, friction, or hydraulic forces acting on the valve control member are not taken into account.

Parameters

- “Basic Parameters Tab” on page 1-404
- “Variable Orifice Tab” on page 1-405
- “Check Valve Tab” on page 1-408

Basic Parameters Tab

Valve pressure setting

The parameter specifies the pressure at port L necessary to start opening the valve. The setting is controlled with the valve spring. The default value is `50e5` Pa.

Valve spring stiffness

The valve spring pressure stiffness c_p. The default value is `1e8` Pa/m.

Pilot ratio

The ratio between the effective areas of the control member face in the pilot chamber and in the load chamber. This is one of the fundamental valve characteristics in a catalog or data sheet. The default value is `3`.

Backpressure ratio

The ratio between the effective area of the control member, onto which the backpressure acts, and the load chamber valve area. There is a wide variety of counterbalance valves with compensated or partially-compensated backpressure. The parameter can take zero value if the valve is completely compensated. The default value is `4`.

Time constant

The valve dynamics are approximated with the first order lag. This parameter is the time constant of the lag. The default value is `0.06` s.

Leakage area

The total area of possible leaks in the completely closed orifice and check valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the orifice or check valve is completely closed. The parameter value must be greater than 0. The default value is `1e-9` m^2.

Variable Orifice Tab

Variable orifice parameterization

Select one of the following methods for specifying the orifice:

- `By maximum area and opening` — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Orifice maximum area

Specify the area of a fully opened orifice. The parameter value must be greater than zero. The default value is `8e-5` m^2. This parameter is used if **Variable orifice parameterization** is set to `By maximum area and opening`.

Orifice maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-4` m. This parameter is used if **Variable orifice parameterization** is set to `By maximum area and opening`.

Orifice flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Orifice laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Orifice laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Orifice critical Reynolds number** parameter.

Orifice laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Orifice laminar transition specification** parameter is set to `Pressure ratio`.

Orifice critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Orifice laminar transition specification** parameter is set to `Reynolds number`.

Tabulated orifice openings

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you

must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-2e-3,0,5e-3,15e-3]`. This parameter is used if **Variable orifice parameterization** is set to `By area vs. opening table`. **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup.

Tabulated orifice area

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in m^2, are `[1e-12,4e-12,1.e-5,1.02e-5]`. This parameter is used if **Variable orifice parameterization** is set to `By area vs. opening table`.

Interpolation method

This parameter is used if **Variable orifice parameterization** is set to `By area vs. opening table`. Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page.

Extrapolation method

This parameter is used if **Variable orifice parameterization** is set to `By area vs. opening table`. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page.

Check Valve Tab

Check valve maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Check valve cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is `3e4` Pa.

Check valve maximum opening pressure

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is `1.2e5` Pa.

Check valve flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Check valve laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Check valve laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Check valve critical Reynolds number** parameter.

Check valve laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Check valve laminar transition specification** parameter is set to `Pressure ratio`.

Check valve critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Check valve laminar transition specification** parameter is set to `Reynolds number`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Variable orifice parameterization**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Variable orifice parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

P

Hydraulic conserving port associated with the valve pilot port.

L

Hydraulic conserving port associated with the valve load port.

B

Hydraulic conserving port associated with the valve backpressure port.

Examples

The Hydraulic Actuator with Counterbalance Valve example illustrates the use of counterbalance valve to balance the load of a double-acting cylinder controlled by an open-center 4-way directional valve.

See Also

Check Valve | Variable Orifice

Introduced in R2012b

Cylinder Cushion

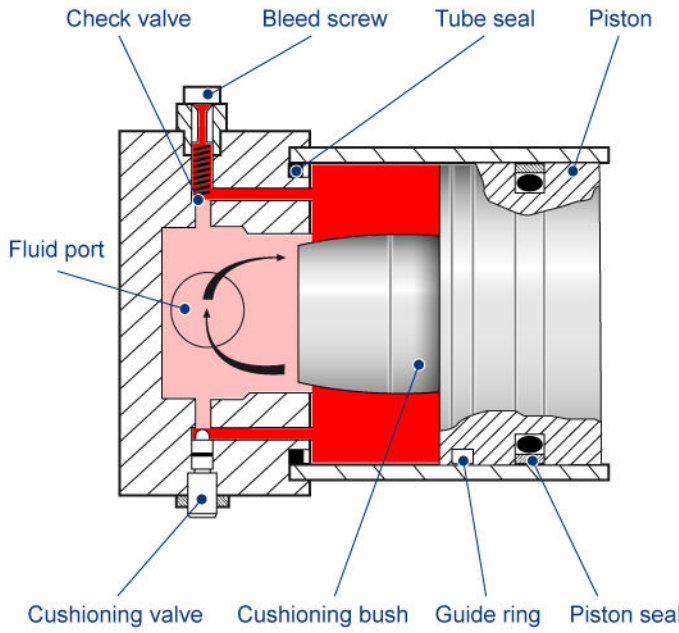
Cushion in hydraulic cylinders

Library

Hydraulic Cylinders

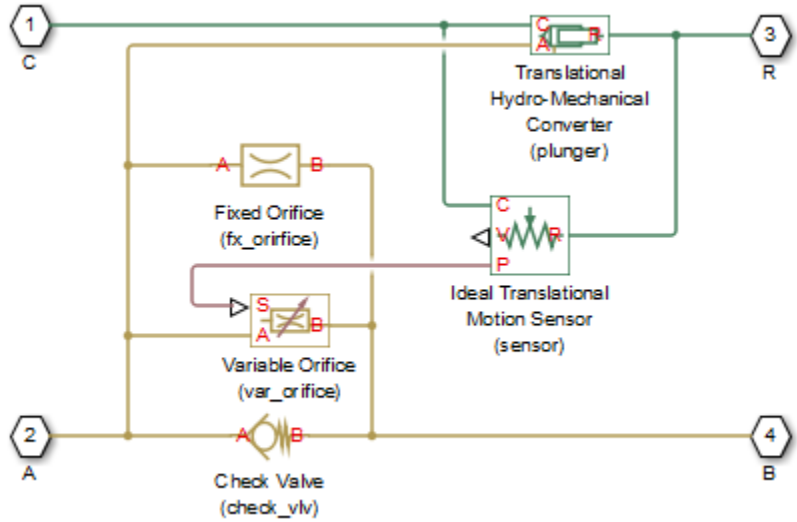
Description

The Cylinder Cushion block models a hydraulic cylinder cushion, the device that decelerates the cylinder rod near the end of the stroke by restricting the flow rate leaving the cylinder chamber. The figure shows a typical design of a cylinder cushion [1] on page 1-421.



As the piston moves toward the cap (to the left in the figure), the cushioning bush enters the chamber in the cap and creates an additional resistance to the fluid leaving the chamber. The bush profile determines the desired deceleration. Near the end of the stroke, the fluid flows through the gap between the bush and the cap and through the cushioning valve with constant cross-sectional area. The check valve located between the chambers allows free flow to the piston chamber to ease the piston breakaway from the end position.

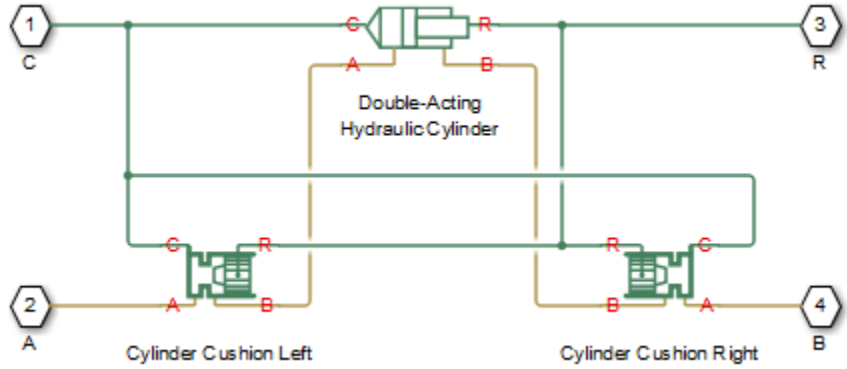
The block is implemented as a structural model that replicates a cushioning device, as shown in this diagram.



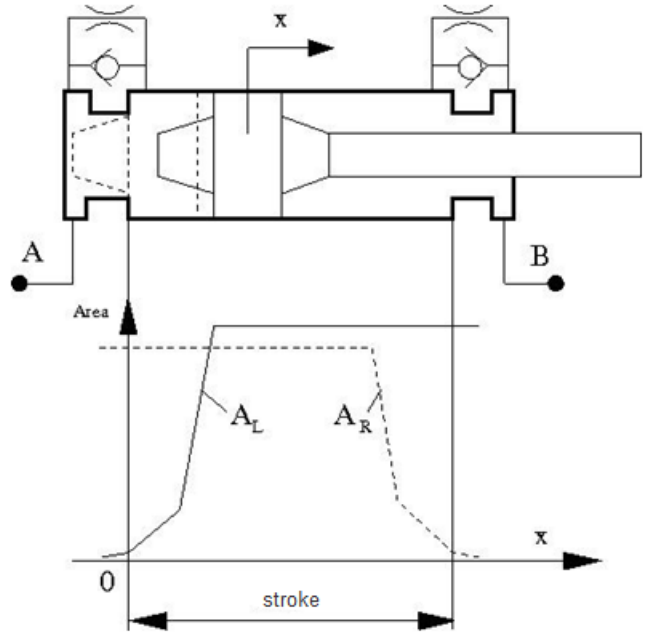
The Variable Orifice block represents a variable gap between the bush and the cavity machined in the end cap. The lookup table of the Variable Orifice block implements the relationship between the orifice area and the piston displacement. The Fixed Orifice and the Check Valve blocks simulate the cushioning valve and the check valve installed between chambers. The Translational Hydro-Mechanical Converter represents a plunger created by the bush and the cavity. The Ideal Translational Motion Sensor block monitors the piston displacement and conveys it (with the initial piston position accounted) to the Variable Orifice block. The names assigned to the nested blocks in the model are shown in parentheses.

The block develops a cushioning effect for the flow rate from port B to port A. The check valve in the block is oriented from port A to port B.

You can use this block with any of the cylinder blocks in the library to model a single-acting or double-acting cylinder with cushion. The following diagram shows the model of a double-sided hydraulic cylinder with cushion built from a Double-Acting Hydraulic Cylinder block and two Cylinder Cushion blocks.

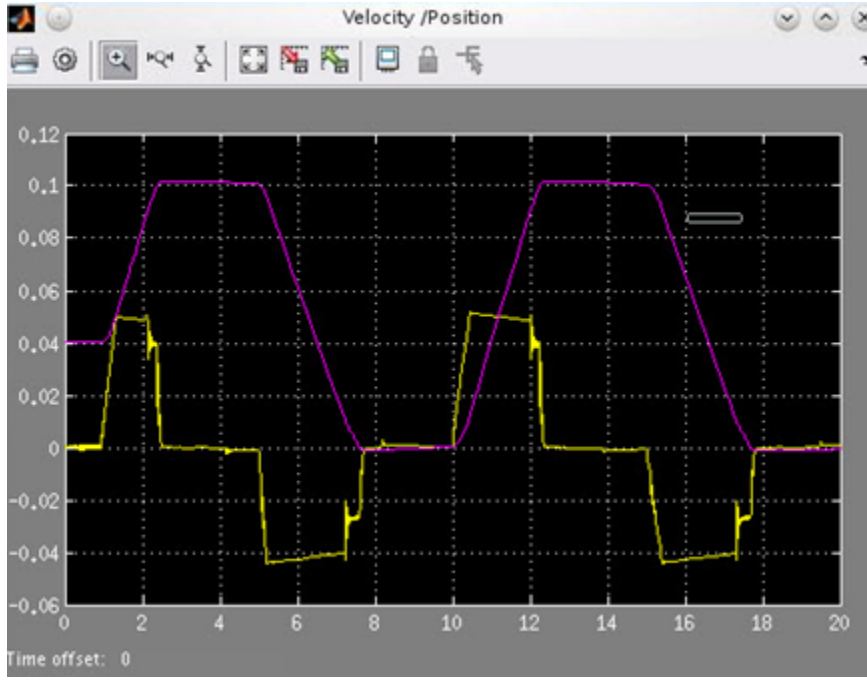


You can adjust the cushioning effect by changing the area of the fixed orifice and the profile of the cushioning bush (variable orifice). Specify the profile using the one-dimensional lookup table of the orifice area versus piston displacement. The next figure shows a typical configuration of a double-acting cylinder with the double-sided cushioning, similar to the model shown in the block diagram above.



To ensure cushioning on both sides of the stroke, set the variable orifice area of the left cushion (A_L) and the right cushion (A_R) similar to the profile shown in the figure. The origin of the plot is located at the position where the piston touches the cap. If the cylinder acts in the negative direction, the piston displacements are negative, and you must make the profile specification in the fourth quadrant.

The following figure shows a typical motion diagram of a cylinder with the double-sided cushioning.



The cushions are set to provide deceleration at ~10 mm before the end of the stroke. The stroke of the cylinder is 10 cm, and the initial position of the piston is 0.04 m. The plot shows the velocity (yellow line) and motion (magenta line) profiles.

Connections A and B are hydraulic conserving ports associated with the device hydraulic inlet and outlet. Connection R is a mechanical translational conserving port that connects to the cylinder rod. Connection C is a mechanical translational conserving port that connects to the cylinder clamping structure.

Parameters

- “Cushion Piston Tab” on page 1-417
- “Fixed Orifice Tab” on page 1-417
- “Variable Orifice Tab” on page 1-418
- “Check Valve Tab” on page 1-419

Cushion Piston Tab

Cushion piston area

Addition of the cushion to a cylinder converts the respective cylinder into two cylinders, rigidly connected and acting in parallel, with the total effective area equal to the piston area before addition. This parameter sets the area of the cushion piston, which is the face area of the cushion bush. The default value is `1e-4` m^2.

Piston initial displacement

The distance between the cylinder piston and cap A at the start of simulation. The default value is `0`, which corresponds to the fully retracted position of the piston.

Piston orientation

Piston orientation with respect to the globally assigned positive direction. Since the cushion piston is part of the cylinder piston, its orientation must be the same as the orientation of the cylinder piston at the side the cushion is attached to. Similar to a cylinder model, if pressure applied at port A exerts force in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Fixed Orifice Tab

Fixed orifice area

The area of the fixed orifice installed between cushion chambers. The default value is `1e-6` m^2.

Fixed orifice flow discharge coefficient

Semi-empirical coefficient that is used in the computation of flow rate through the fixed orifice. The default value is `0.7`.

Fixed orifice critical Reynolds number

The maximum Reynolds number for laminar flow through the fixed orifice. The transition from laminar to turbulent regime is supposed to take place when the Reynolds number reaches this value. The value of the parameter depends on the orifice geometrical profile. You can find the recommendations on the parameter value in hydraulic textbooks. The default value is `10`.

Variable Orifice Tab

Tabulated piston displacements

Vector of input values for piston displacements, specified as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The **Tabulated piston displacements** values are used together with **Tabulated orifice area** values for one-dimensional table lookup. Due to the nature of the cylinder hard stops, the piston can move below zero and above the stroke value. It is good practice to account for piston deformation and provide piston displacements beyond the ideal stroke range to avoid extrapolation. The default values, in mm, are `[-2, 10, 11, 89, 90, 102]`.

Tabulated orifice area

Vector of orifice areas, specified as a one-dimensional array. The vector must be the same size as the piston displacements vector. All the values must be positive. The default values, in cm^2, are `[0.004, 0.006, 9, 9, 9, 9]`.

Variable orifice interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page.

Variable orifice extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page.

Variable orifice flow discharge coefficient

Semi-empirical coefficient that is used in the computation of flow rate through the variable orifice. The default value is `0.7`.

Variable orifice critical Reynolds number

The maximum Reynolds number for laminar flow through the variable orifice. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `10`.

Variable orifice leakage area

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Check Valve Tab

Check valve maximum area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Check valve cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is `0.5e5` Pa.

Check valve full opening pressure

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is `1.5e5` Pa.

Check valve flow discharge coefficient

Semi-empirical coefficient that is used in the computation of flow rate through the check valve. The default value is `0.7`.

Check valve critical Reynolds number

The maximum Reynolds number for laminar flow through the check valve. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is `10`.

Check valve leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12 m^2`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Piston orientation**
- **Variable orifice interpolation method**
- **Variable orifice extrapolation method**

All other block parameters are available for modification.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port connected to the cylinder inlet.

`B`

Hydraulic conserving port connected to the cylinder outlet.

`R`

Mechanical translational conserving port connected to the cylinder rod.

`C`

Mechanical translational conserving port connected to the cylinder clamping structure.

References

[1] Rohner, P. *Industrial Hydraulic Control*. Fourth edition. Brisbane : John Wiley & Sons, 1995.

See Also

Check Valve | Fixed Orifice | Variable Orifice

Introduced in R2013b

Cylinder Friction

Friction in hydraulic cylinders

Library

Hydraulic Cylinders

Description

The Cylinder Friction block simulates friction in the contact between moving bodies in hydraulic cylinders and is intended to be used primarily as a building block in combination with both the double- and single-acting cylinders to develop a cylinder model with friction. The friction force is simulated as a function of relative velocity and pressure, and is assumed to be the sum of Stribeck, Coulomb, and viscous components. The Coulomb friction force consists of the preload force, caused by the seal squeeze during assembly, and the force proportional to pressure. The sum of the Coulomb and Stribeck friction forces at zero velocity is often referred to as the breakaway friction force. For more information, see the Translational Friction block reference page.

The friction force is approximated with the following equations:

$$F = F_C \cdot \left(1 + \left(K_{brk} - 1\right) \cdot \exp\left(-c_v \left| v \right|\right)\right) sign\left(v\right) + f_{vfr} \cdot v$$

$$F_C = F_{pr} + f_{cfr}\left(p_A + p_B\right)$$

where

F	Friction force

F_C	Coulomb friction
F_{pr}	Preload force
f_{cfr}	Coulomb friction coefficient
p_A, p_B	Pressures in cylinder chambers
K_{brk}	Breakaway friction force increase coefficient
c_v	Transition coefficient
v	Relative velocity in the contact
f_{vfr}	Viscous friction coefficient

To avoid discontinuity at $v = 0$, a small region $|v| \le v_{th}$ is introduced around zero velocity, where friction force is assumed to be linearly proportional to velocity:

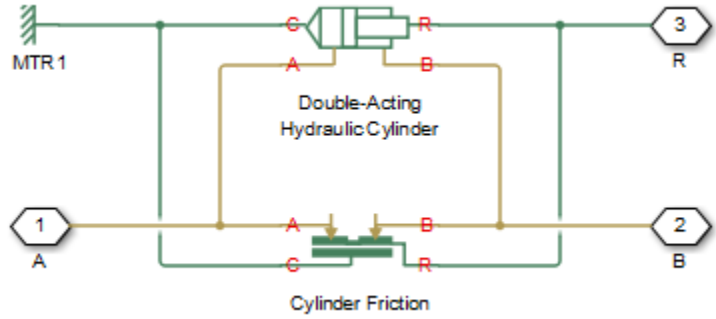
$$F = K \cdot v$$

$$K = \frac{F_C\left(1 + \left(K_{brk} - 1\right) \cdot \exp\left(-c_v v_{th}\right)\right) + f_{vfr} \cdot v_{th}}{v_{th}}$$

where

K	Proportionality coefficient
v_{th}	Velocity threshold

Connections R and C are mechanical translational conserving ports associated with the rod and case, respectively. Connections A and B are hydraulic conserving ports to be connected to ports A and B of the cylinder model, as shown in the following illustration. The force generated by the block always opposes relative motion between the rod and the case.



Parameters

- “Parameters Tab” on page 1-424
- “Variables Tab” on page 1-425

Parameters Tab

Preload force

The preload force, caused by the seal squeeze during assembly. The default value is `10` N.

Coulomb friction force coefficient

Coulomb friction coefficient, which defines the proportionality between the Coulomb friction force and the pressure in cylinder chambers. The default value is `1e-6` N/Pa.

Breakaway friction increase coefficient

The friction force increase over the Coulomb friction. The Coulomb friction force, multiplied by this coefficient, is referred to as breakaway friction force. The default value is `1`.

Viscous friction coefficient

Proportionality coefficient between the viscous friction force and the relative velocity. The parameter value must be greater than or equal to zero. The default value is `100` N/(m/s).

Transition approximation coefficient

The parameter sets the value of coefficient c_v, which is used for the approximation of the transition between the breakaway and the Coulomb frictions. Its value is assigned

based on the following considerations: the Stribeck friction component reaches approximately 5% of its steady-state value at velocity $3/c_v$, and 2% at velocity $4/c_v$, which makes it possible to develop an approximate relationship c_v ~= $4/v_{min}$, where v_{min} is the relative velocity at which friction force has its minimum value. By default, c_v is set to `10` s/m, which corresponds to a minimum friction at velocity of about `0.4` m/s.

Linear region velocity threshold

The parameter sets the small vicinity near zero velocity, within which friction force is considered to be linearly proportional to the relative velocity. MathWorks recommends that you use values in the range between `1e-6` and `1e-4` m/s. The default value is `1e-4` m/s.

Variables Tab

Use the **Variables** tab to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Ports

The block has the following ports:

`A`

Hydraulic conserving port connected to the cylinder inlet.

`B`

Hydraulic conserving port connected to the cylinder outlet.

`R`

Mechanical translational conserving port associated with the cylinder rod.

`C`

Mechanical translational conserving port associated with the cylinder clamping structure.

See Also

Double-Acting Hydraulic Cylinder | Single-Acting Hydraulic Cylinder

Introduced in R2006b

Double-Acting Actuator (TL)

Linear actuator with piston motion controlled by two opposing thermal liquid chambers

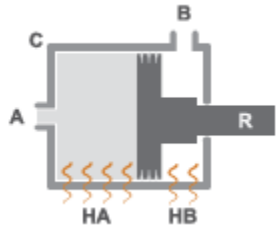
Library

Thermal Liquid/Actuators

Description

The Double-Acting Actuator (TL) block models a linear actuator with piston motion controlled by two opposing thermal liquid chambers. The actuator generates force in the extension and retraction strokes. The force generated depends on the pressure difference between the two chambers.

The figure shows the key components of the actuator model. Ports A and B represent the thermal liquid chamber inlets. Port R represents the translating actuator piston and port C the actuator case. Ports HA and HB represent the thermal interfaces between each chamber and the environment. The moving piston is assumed perfectly insulating.



Double-Acting Actuator Schematic

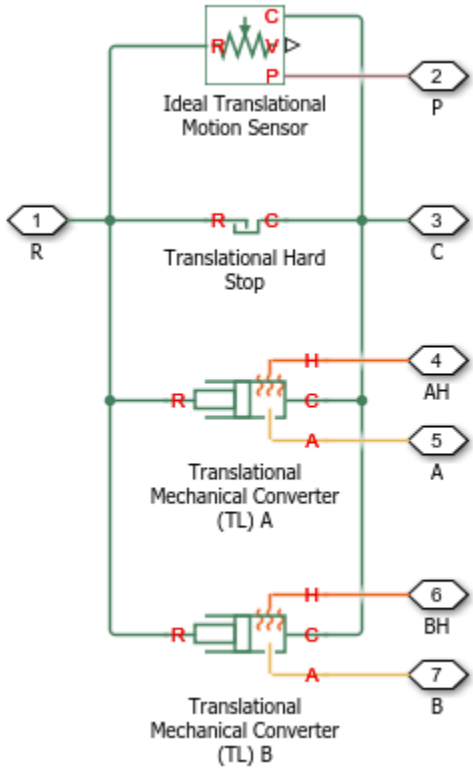
The direction of the piston motion depends on the mechanical orientation setting in the block dialog box. If the mechanical orientation is positive, then a higher pressure at port A yields a positive piston translation relative to the actuator case. The direction of motion reverses for a negative mechanical orientation.

A set of hard stops limit the piston range of motion. The hard stops are treated as spring-damper systems. The spring stiffness coefficient controls the restorative component of the hard-stop contact force and the damping coefficient the dissipative component.

The hard stops are located at the distal ends of the piston stroke. If the mechanical orientation is positive, then the lower hard stop is at $x = 0$ and the upper hard stop at $x = +stroke$. If the mechanical orientation is negative, then the lower hard stop is at $x = -stroke$ and the upper hard stop at $x = 0$.

This block is a composite component based on the Simscape Foundation blocks:

- Translational Mechanical Converter (TL)
- Translational Hard Stop
- Ideal Translational Motion Sensor



Composite Component Diagram

Parameters

Actuator Tab

Mechanical orientation

Orientation of the actuator piston relative to the direction of flow. A positive orientation causes the piston to move in the positive direction relative to the actuator casing in response to a positive flow rate through port A. The default setting is `Positive`.

The mechanical orientation affects the placement of the actuator hard stops. One hard stop is always at position zero. The second hard stop is at the piston stroke distance if the mechanical orientation is positive and at minus the piston stroke distance if the mechanical orientation is negative.

Piston cross-sectional area at A

Area normal to the direction of flow in actuator chamber A. The block uses this area to calculate the hydraulic force due to the fluid pressure in chamber A. The piston cross-sectional area must be greater than zero. The default value is `0.01` m^2.

Piston cross-sectional area at B

Area normal to the direction of flow in actuator chamber B. The block uses this area to calculate the hydraulic force due to the fluid pressure in chamber B. The piston cross-sectional area must be greater than zero. The default value is `0.01` m^2.

Piston stroke

Maximum distance the actuator piston can travel. The piston stroke must be greater than zero. The default value is `0.1` m.

Hard stops limit piston motion to the length of the piston stroke. One hard stop is located at position zero. The second hard stop is at the piston stroke distance if **Mechanical Orientation** is set to `Positive` and at minus the piston stroke if **Mechanical Orientation** is set to `Negative`.

Dead volume at A

Fluid volume remaining in actuator chamber A at a zero piston displacement. The block uses this volume to account for mass and energy storage in chamber A when the piston is at position zero. The dead volume must be greater than zero. The default value is `1e-5` m^3.

Dead volume at B

Fluid volume remaining in actuator chamber B at a zero piston displacement. The block uses this volume to account for mass and energy storage in chamber B when the piston is at position zero. The dead volume must be greater than zero. The default value is `1e-5` m^3.

Environment pressure specification

Choice of environment pressure. Options include `Atmospheric pressure` and `Specified pressure`. Selecting `Specified pressure` exposes an additional parameter, **Environment pressure**.

Environment pressure

Pressure outside the actuator casing. This pressure acts against the pressures inside the actuator chambers. A value of zero corresponds to a vacuum. The default value is

`0.101325` MPa. This parameter is visible only when **Environment pressure specification** is set to `Specified pressure`.

Hard Stop Tab

Hard-stop stiffness coefficient

Spring coefficient of the actuator hard stops. The spring coefficient accounts for the restorative portion of the hard-stop contact force. Increase the coefficient value to model harder contact. The default value is `1e10` N/m.

Hard-stop damping coefficient

Damping coefficient of the actuator hard stops. The damping coefficient accounts for the dissipative portion of the hard-stop contact force. Increase the coefficient value to reduce bounce upon contact. The default value is `150` N/(m/s).

Hard stop model

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` (default) — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.
- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

Transition region

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero. The default value is `1mm`.

Initial Conditions Tab

Piston initial displacement

Piston position at the start of simulation. This value must be between zero and the piston stroke if the **Mechanical orientation** parameter is set to `Positive`. It must be between zero and minus the piston stroke if the **Mechanical orientation** parameter is set to `Negative`. The default value is `0` m.

Initial liquid temperature at A

Temperature in actuator chamber A at the start of simulation. The default value is `293.15` K.

Initial liquid temperature at B

Temperature in actuator chamber B at the start of simulation. The default value is `293.15` K.

Fluid dynamic compressibility

Option to model effects due to fluid dynamic compressibility. Select `On` to enable fluid dynamic compressibility and `Off` to disable it.

Initial liquid pressure in chamber A

Pressure in actuator chamber A at the start of simulation. The default value is `0.101325` MPa.

Initial liquid pressure in chamber B

Pressure in actuator chamber B at the start of simulation. The default value is `0.101325` MPa.

Ports

- A — Thermal liquid conserving port representing actuator chamber A
- B — Thermal liquid conserving port representing actuator chamber B
- C — Mechanical conserving port representing the actuator case
- R — Mechanical conserving port representing the actuator piston
- HA — Thermal conserving port representing the thermal interface between chamber A and the environment
- HB — Thermal conserving port representing the thermal interface between chamber B and the environment

- P — Physical signal output port for the piston position data

See Also

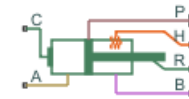
Single-Acting Actuator (TL)

Introduced in R2016a

Double-Acting Actuator (IL-G)

Linear actuator with piston motion controlled by opposing hydraulic and gas chambers

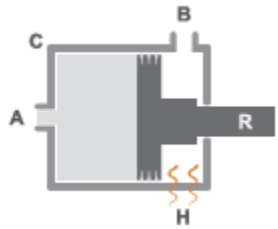
Library: Simscape / Fluids / Fluid Network Interfaces / Actuators



Description

The Double-Acting Actuator (IL-G) block represents a linear actuator with piston motion controlled by two opposing fluid chambers, one filled with gas, the other with hydraulic (isothermal liquid) fluid. The actuator generates force in the extension and retraction strokes. The force generated depends on the pressure difference between the two chambers. This block constitutes an interface between four Simscape domains—Gas, Hydraulic, Thermal, and Mechanical Translational.

The figure shows the key components of the actuator model. Ports A and B represent the inlets to the hydraulic fluid and gas chambers, respectively. Port R represents the translating actuator piston and port C the actuator case. Port H represents the thermal interface between the gas chamber and the environment. The temperature of the hydraulic fluid is assumed constant and any heat exchange with the environment is ignored. The piston is assumed perfectly insulating.



The direction of motion of the piston depends on the mechanical orientation specified in the block dialog box. If the mechanical orientation is positive, then a higher pressure at port A yields a positive piston translation relative to the actuator casing. If the mechanical

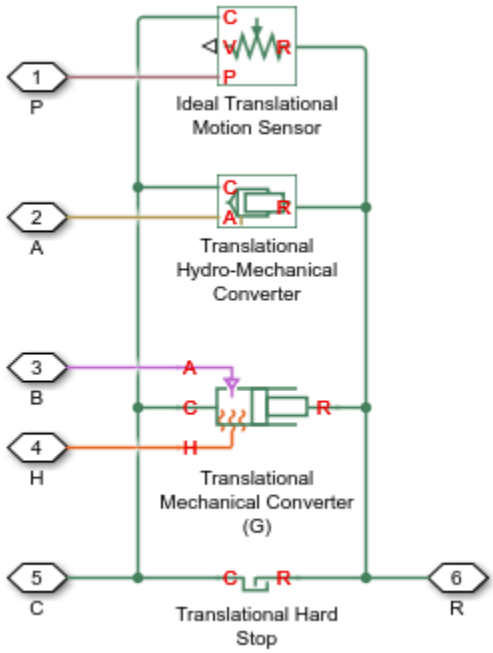
orientation is negative, then a higher pressure at port A yields a negative piston translation instead.

A set of hard stops limit the piston range of motion. The hard stops are treated as spring-damper systems. The spring stiffness coefficient controls the restorative component of the hard-stop contact force. The damping coefficient controls the dissipative component of the force. The hard stops are located at the distal ends of the piston stroke, with the exact locations partly dependent on the piston stroke (d_s) and piston initial distance from the hydraulic inlet (x_0):

- If the **Mechanical orientation** parameter is set to `Positive`, then the lower hard stop is at $-x_0$ and the upper hard stop at $d_s - x_0$.
- If the **Mechanical orientation** parameter is set to `Negative`, then the lower hard stop is at $-d_s + x_0$ and the upper hard stop at x_0.

The block is a composite component built from Simscape Foundation blocks. For more information on how the Double-Acting Actuator (IL-G) block works, see the reference pages of the constituent blocks:

- Translational Hydro-Mechanical Converter
- Translational Mechanical Converter (G)
- Translational Hard Stop
- Ideal Translational Motion Sensor



Ports

Output

P — Piston position, m

physical signal

Physical signal output port for the piston position data.

Conserving

A — Inlet to hydraulic chamber

hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the inlet to the hydraulic chamber of the actuator.

B — Inlet to gas chamber
gas

Gas conserving port representing the inlet to the gas chamber of the actuator.

R — Actuator piston
mechanical translational

Mechanical translational conserving port representing the actuator piston. The piston is capable of translational motion relative to the casing.

C — Actuator casing
mechanical translational

Mechanical translational conserving port representing the actuator casing. The casing serves as a mechanical reference for the motion of the piston.

H — Gas chamber thermal surface
Thermal

Thermal conserving port representing the surface of the gas chamber through which heat exchange with the surroundings can occur. The thermal processes at this port determine the temperature in the gas chamber and therefore at port B.

Parameters

Hydraulic Side

`Mechanical orientation` — Direction of motion of the piston relative to the direction of flow of the fluid
`Positive` | `Negative`

Orientation of the actuator piston relative to the direction of fluid flow. A positive orientation causes the piston to move in the positive direction relative to the actuator casing in response to a positive flow rate through port A. The mechanical orientation affects the placement of the piston hard stops. See the block description for more information on the hard stop placement.

`Piston cross-sectional area in chamber A` — Cross-sectional area of the fluid volume in the hydraulic chamber
`2e-3 m^2` (default) | `Scalar in units of area`

Area normal to the direction of flow in the body of the hydraulic chamber. The block uses this area to calculate the hydraulic force due to the fluid pressure in the hydraulic chamber. This parameter must be greater than zero.

`Piston stroke` — Distance of travel of piston from hard stop to hard stop
`0.2 m` (default) | `Scalar in units of length`

Total distance of travel available to the piston, from one hard stop to the other. The hard stops limit the piston motion so that the piston is confined to stroke of the piston. See the block description for more information on the locations of the hard stops.

`Piston initial distance from port A` — Piston position relative to the hydraulic chamber inlet at the start of simulation
`0 m` (default) | `Scalar in units of length`

Distance of the piston from the hard stop closest to the hydraulic inlet at the start of simulation. Use this parameter to change the starting position of the piston. This parameter affects the placement of the piston hard stops. For more information on the hard stop placement, see the block description.

`Compressibility` — Choice to model the effects of dynamic compressibility inside the hydraulic chamber
`Off` (default) | `On`

Option to model the effects of dynamic compressibility inside the hydraulic chamber. The hydraulic fluid is treated as compressible if this parameter is set to `On` and as incompressible if it is set to `Off`. The block ignores the dependence of the hydraulic fluid density on pressure and temperature if `Off` is selected.

`Dead volume in chamber A` — Fluid volume in the hydraulic chamber in the bottomed-out position
`1e-4 m^3` (default) | Scalar in units of volume

Hydraulic fluid volume remaining in the hydraulic chamber when the piston is pressed against the hard stop closest to the hydraulic inlet. The dead volume enables the block to capture the internal states of the hydraulic fluid volume—its pressure and temperature—when this volume is at a minimum. This parameter must be greater than zero.

Dependencies

This parameter is enabled when the **Compressibility** parameter is set to `On`.

`Specific heat ratio` — Specific heat ratio of the gas entrained in the hydraulic fluid

`1.4` (default) | Unitless scalar

Ratio of the specific heat of the gas entrained in the hydraulic fluid at constant pressure to that at constant volume. The block uses this parameter in the calculations of density for the hydraulic fluid.

Dependencies

This parameter is enabled when the **Compressibility** parameter is set to `On`.

`Initial liquid pressure (relative)` — Gauge pressure in the hydraulic chamber relative to the actuator environment

`0 Pa` (default) | Scalar in units of pressure

Pressure inside the hydraulic chamber at simulation time zero relative to absolute zero. This parameter helps set the initial states of the hydraulic fluid volume.

Dependencies

This parameter is enabled when the **Compressibility** parameter is set to `On`.

Gas Side

`Piston cross-sectional area in chamber B` — Cross-sectional area of the fluid volume in the gas chamber

`1e-3 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow in the body of the gas chamber. The block uses this area to calculate the pneumatic force due to the fluid pressure in the gas chamber. This parameter must be greater than zero.

`Cross-sectional area at port B` — Cross-sectional area of the inlet of the gas chamber

`1e-3 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow at the entrance to the gas chamber. The cross-sectional area at the entrance can differ from that in the body of the chamber. Set the two cross-sectional areas to different values to model the effects of a sudden area change at the inlet. This parameter must be greater than zero.

`Dead volume in chamber B` — Fluid volume in the gas chamber in the bottomed-out position
`1e-4 m^3` (default) | positive scalar in units of volume

Gas volume remaining in the gas chamber when the piston is pressed against the hard stop closest to the gas inlet. The dead volume enables the block to capture the internal states of the gas volume—its pressure and temperature—when this volume is at a minimum. This parameter must be greater than zero.

`Initial gas pressure (absolute)` — Absolute pressure inside the gas chamber at the start of simulation
`0.101325 MPa` (default) | scalar in units of pressure

Pressure inside the gas chamber at simulation time zero relative to absolute zero. This parameter helps set the initial state of the gas volume.

`Environment pressure specification` — Choice of atmospheric or custom environment pressure for the gas side
`Atmospheric pressure` (default) | `Specified pressure`

Option to set the environment pressure of the gas chamber to the typical value of one earth atmosphere or to a custom value. Selecting `Specified pressure` exposes an additional parameter, **Environment pressure**, that you use to specify a custom pressure.

`Environment pressure` — Absolute pressure of the environment outside the gas chamber
`0.101325 MPa` (default) | scalar in units of pressure

Pressure outside the gas chamber relative to absolute zero. This pressure acts against the pressure inside the gas chamber. A pressure of zero corresponds to a perfect vacuum.

Dependencies

This parameter is enabled when the **Environment pressure specification** is set to `Specified pressure`.

Hard Stop

`Hard-stop stiffness coefficient` — Stiffness coefficient for the restorative portion of the hard-stop contact force
`1e6 N/m` (default) | scalar in units of force/length

Spring coefficient for use in the spring-damper model of the piston hard-stops. The spring force is assumed to be the same at both hard stops. Increase the coefficient value to model harder contact.

`Hard-stop damping coefficient` — Damping coefficient for the dissipative portion of the hard-stop contact force
`150 N/(m/s)` (default) | scalar in units of force/velocity

Damping coefficient for use in the spring-damper model of the piston hard stops. The damping force is assumed to be the same at both hard stops. Increase the coefficient value to reduce piston bounce on contact.

`Hard stop model` — Modeling approach for hard stops
`Stiffness and damping applied smoothly through transition region` (default) | Full stiffness and damping applied at bounds, undamped rebound | Full stiffness and damping applied at bounds, damped rebound

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.
- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

`Transition region` — Distance below which scaling is applied to the hard-stop force
`0.1 mm` (default) | scalar in units of length

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero.

See Also

Double-Acting Actuator (TL) | Double-Acting Actuator (TL-G) | Double-Acting Hydraulic Cylinder | Double-Acting Hydraulic Cylinder (Simple)

Introduced in R2016b

Double-Acting Actuator (TL-G)

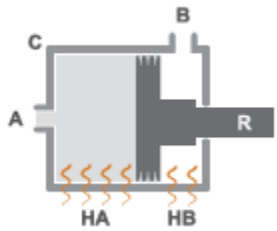
Linear actuator with piston motion controlled by opposing thermal liquid and gas chambers

Library: Simscape / Fluids / Fluid Network Interfaces / Actuators

Description

The Double-Acting Actuator (TL-G) block represents a linear actuator with piston motion controlled by two opposing fluid chambers, one filled with gas, the other with thermal liquid. The actuator generates force in the extension and retraction strokes. The force generated depends on the pressure difference between the two chambers. This block constitutes an interface between four Simscape domains—Gas, Thermal Liquid, Thermal, and Mechanical Translational.

The figure shows the key components of the actuator model. Ports A and B represent the inlets to the thermal liquid and gas chambers, respectively. Port R represents the translating actuator piston and port C the actuator case. Ports HA and HB represent the thermal interfaces between each chamber and the environment. The piston is assumed perfectly insulating.



The direction of motion of the piston depends on the mechanical orientation specified in the block dialog box. If the mechanical orientation is positive, then a higher pressure at port A yields a positive piston translation relative to the actuator casing. If the mechanical

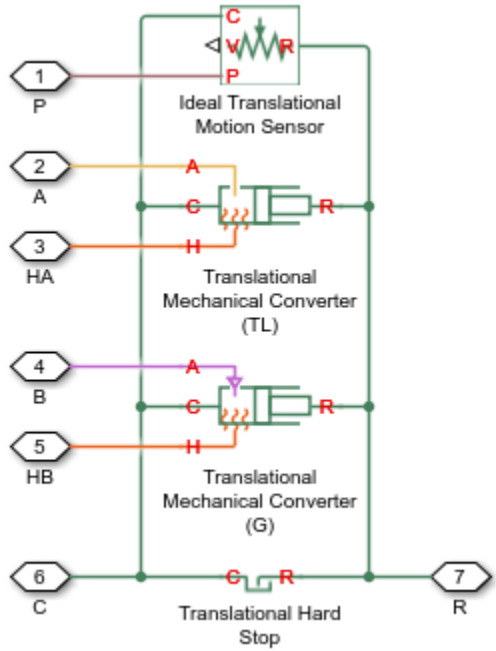
orientation is negative, then a higher pressure at port A yields a negative piston translation instead.

A set of hard stops limit the piston range of motion. The hard stops are treated as spring-damper systems. The spring stiffness coefficient controls the restorative component of the hard-stop contact force. The damping coefficient controls the dissipative component of the force. The hard stops are located at the distal ends of the piston stroke, with the exact locations dependent on the piston stroke (d_s):

- If the **Mechanical orientation** parameter is set to `Positive`, then the lower hard stop is at 0 and the upper hard stop at d_s.
- If the **Mechanical orientation** parameter is set to `Negative`, then the lower hard stop is at $-d_s$ and the upper hard stop at 0.

The block is a composite component built from Simscape Foundation blocks. For more information on how the Double-Acting Actuator (TL-G) block works, see the reference pages of the constituent blocks:

- Translational Mechanical Converter (TL)
- Translational Mechanical Converter (G)
- Translational Hard Stop
- Ideal Translational Motion Sensor



Ports

Output

P — Piston position, m

physical signal

Physical signal output port for the piston position data.

Conserving

A — Inlet to thermal liquid chamber

hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the inlet to the hydraulic chamber of the actuator.

B — Inlet to gas chamber
gas

Gas conserving port representing the inlet to the gas chamber of the actuator.

R — Actuator piston
mechanical translational

Mechanical translational conserving port representing the actuator piston. The piston is capable of translational motion relative to the casing.

C — Actuator casing
mechanical translational

Mechanical translational conserving port representing the actuator casing. The casing serves as a mechanical reference for the motion of the piston.

HA — Heat exchange surface of the thermal liquid chamber
Thermal

Thermal conserving port representing the surface through which heat exchange can occur between the thermal liquid volume and the actuator surroundings. The thermal processes at this port influence the temperature in the thermal liquid chamber and therefore at port A.

HB — Heat exchange surface of the gas chamber
Thermal

Thermal conserving port representing the surface through which heat exchange can occur between the gas volume and the actuator surroundings. The thermal processes at this port influence the temperature in the gas chamber and therefore at port A.

Parameters

Thermal liquid Side

`Mechanical orientation` — Direction of motion of the piston relative to the direction of flow of the fluid
`Positive` | `Negative`

Orientation of the actuator piston relative to the direction of fluid flow. A positive orientation causes the piston to move in the positive direction relative to the actuator

casing in response to a positive flow rate through port A. The mechanical orientation affects the placement of the piston hard stops. See the block description for more information on the hard stop placement.

`Piston cross-sectional area in chamber A` — Cross-sectional area of the fluid volume in the thermal liquid chamber
`0.01 m^2` (default) | scalar in units of area

Area normal to the direction of flow in the body of the thermal liquid chamber. The block uses this area to calculate the hydraulic force due to the fluid pressure in the thermal liquid chamber. This parameter must be greater than zero.

`Piston stroke` — Distance of travel of the piston from hard stop to hard stop
`0.2 m` (default) | scalar in units of length

Total distance of travel available to the piston, from one hard stop to the other. The hard stops limit the piston motion so that the piston is confined to stroke of the piston. See the block description for more information on the locations of the hard stops.

`Piston initial displacement` — Position of the piston at the start of simulation
`0 m` (default) | scalar in units of length

Distance of the piston from the hard stop closest to the thermal liquid inlet at the start of simulation. Use this parameter to change the starting position of the piston. This parameter has no impact on the placement of the piston hard stops.

`Dead volume in chamber A` — Fluid volume in the thermal liquid chamber in the bottomed-out position
`1e-4 m^3` (default) | scalar in units of volume

Thermal liquid volume remaining in the thermal liquid chamber when the piston is pressed against the hard stop closest to the thermal liquid inlet. The dead volume enables the block to capture the internal states of the thermal liquid volume—its pressure and temperature—when this volume is at a minimum. This parameter must be greater than zero.

`Fluid dynamic compressibility` — Choice to model the effects of dynamic compressibility inside the thermal liquid chamber
`On` (default) | `Off`

Option to model the effects of dynamic compressibility inside the thermal liquid chamber. The thermal liquid is treated as compressible if this parameter is set to `On` and as

incompressible if it is set to `Off`. The block ignores the dependence of the thermal liquid density on pressure and temperature if `Off` is selected.

`Initial liquid pressure in chamber A` — Absolute pressure in the thermal liquid chamber
`0.101325 Pa` (default) | scalar in units of pressure

Pressure inside the thermal liquid chamber at simulation time zero relative to absolute zero. This parameter helps set the initial states of the thermal liquid volume.

Dependencies

This parameter is enabled when the **Compressibility** parameter is set to `On`.

`Initial liquid temperature in chamber A` — Temperature in the thermal liquid chamber at the start of simulation
`293.15 K` (default) | scalar in units of temperature

Average temperature inside the thermal liquid chamber at the start of simulation. This parameter helps set the initial states of the thermal liquid volume.

`Environment pressure specification` — Choice of atmospheric or custom environment pressure for the thermal liquid side
`Atmospheric pressure` (default) | `Specified pressure`

Option to set the environment pressure of the thermal liquid chamber to the typical value of one earth atmosphere or to a custom value. Selecting `Specified pressure` exposes an additional parameter, **Environment pressure**, that you use to specify a custom pressure.

`Environment pressure` — Absolute pressure of the environment outside the thermal liquid chamber
`0.101325 MPa` (default) | scalar in units of pressure

Pressure outside the thermal liquid chamber relative to absolute zero. This pressure acts against the pressure inside the thermal liquid chamber. A pressure of zero corresponds to a perfect vacuum.

Dependencies

This parameter is enabled when the **Environment pressure specification** is set to `Specified pressure`.

Gas Side

`Piston cross-sectional area in chamber B` — Cross-sectional area of the fluid volume in the gas chamber
`1e-3 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow in the body of the gas chamber. The block uses this area to calculate the pneumatic force due to the fluid pressure in the gas chamber. This parameter must be greater than zero.

`Cross-sectional area at port B` — Cross-sectional area of the inlet of the gas chamber
`1e-3 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow at the entrance to the gas chamber. The cross-sectional area at the entrance can differ from that in the body of the chamber. Set the two cross-sectional areas to different values to model the effects of a sudden area change at the inlet. This parameter must be greater than zero.

`Dead volume in chamber B` — Fluid volume in the gas chamber in the bottomed-out state
`1e-4 m^3` (default) | positive scalar in units of volume

Gas volume remaining in the gas chamber when the piston is pressed against the hard stop closest to the gas inlet. The dead volume enables the block to capture the internal states of the gas volume—its pressure and temperature—when this volume is at a minimum. This parameter must be greater than zero.

`Initial gas pressure (absolute)` — Absolute pressure inside the gas chamber at the start of simulation
`0.101325 MPa` (default) | scalar in units of pressure

Pressure inside the gas chamber at simulation time zero relative to absolute zero. This pressure helps set the initial state of the gas volume.

`Environment pressure specification` — Choice of atmospheric or custom environment pressure for the gas side
`Atmospheric pressure` (default) | `Specified pressure`

Option to set the environment pressure of the gas chamber to the typical value of one earth atmosphere or to a custom value. Selecting `Specified pressure` exposes an additional parameter, **Environment pressure**, that you use to specify a custom pressure.

`Environment pressure` — Absolute pressure of the environment outside the gas chamber
`0.101325 MPa` (default) | scalar in units of pressure

Pressure outside the gas chamber relative to absolute zero. This pressure acts against the pressure inside the gas chamber. A pressure of zero corresponds to a perfect vacuum.

Dependencies

This parameter is enabled when the **Environment pressure specification** is set to `Specified pressure`.

Hard Stop

`Hard-stop stiffness coefficient` — Stiffness coefficient for the restorative portion of the hard-stop contact force
`1e6 N/m` (default) | scalar in units of force/length

Spring coefficient for use in the spring-damper model of the piston hard-stops. The spring force is assumed to be the same at both hard stops. Increase the coefficient value to model harder contact.

`Hard-stop damping coefficient` — Damping coefficient for the dissipative portion of the hard-stop contact force
`150 N/(m/s)` (default) | scalar in units of force/velocity

Damping coefficient for use in the spring-damper model of the piston hard stops. The damping force is assumed to be the same at both hard stops. Increase the coefficient value to reduce piston bounce on contact.

`Hard stop model` — Modeling approach for hard stops
`Stiffness and damping applied smoothly through transition region` (default) | Full stiffness and damping applied at bounds, undamped rebound | Full stiffness and damping applied at bounds, damped rebound

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is

breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.

- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

`Transition region` — Distance below which scaling is applied to the hard-stop force

`0.1 mm` (default) | scalar in units of length

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero.

See Also

Double-Acting Actuator (IL-G) | Double-Acting Actuator (TL) | Double-Acting Hydraulic Cylinder | Double-Acting Hydraulic Cylinder (Simple)

Introduced in R2016b

Double-Acting Hydraulic Cylinder

Hydraulic actuator exerting force in both directions

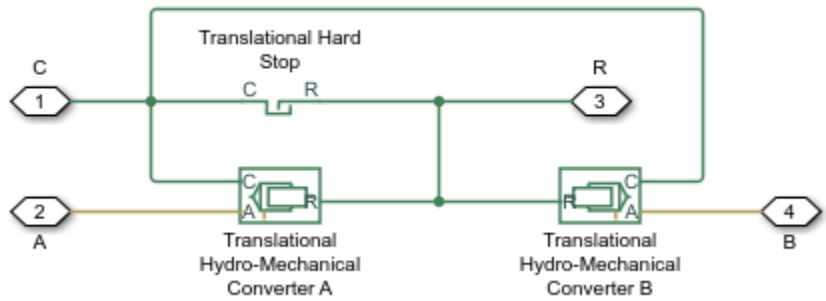
Library

Hydraulic Cylinders

Description

The Double-Acting Hydraulic Cylinder block models a device that converts hydraulic energy into mechanical energy in the form of translational motion. Hydraulic fluid pumped under pressure into one of the two cylinder chambers forces the piston to move and exert force on the cylinder rod. Double-acting cylinders transfer force and motion in both directions.

The model of the cylinder is built of Simscape Foundation library blocks. The schematic diagram of the model is shown below.



Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connections A and B are

hydraulic conserving ports. Port A is connected to converter A and port B is connected to converter B.

The energy through hydraulic port A or B is directed to the appropriate Translational Hydro-Mechanical Converter block. The converter transforms hydraulic energy into mechanical energy and accounts for the fluid compressibility in the cylinder chamber. The rod motion is limited with the mechanical Translational Hard Stop block in such a way that the rod can travel only between cylinder caps.

The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

Basic Assumptions and Limitations

- No leakage, internal or external, is taken into account.
- No loading on piston rod, such as inertia, friction, spring, and so on, is taken into account. If necessary, you can easily add them by connecting an appropriate building block to cylinder port R.

Parameters

- “Basic Parameters Tab” on page 1-453
- “Hard Stop Properties Tab” on page 1-454
- “Initial Conditions Tab” on page 1-455

Basic Parameters Tab

Piston area A

Chamber A effective piston area. The default value is `0.001` m^2.

Piston area B

Chamber B effective piston area. The default value is `5e-5` m^2.

Piston stroke

Piston maximum travel between caps. The default value is `0.1` m.

Dead volume A

Fluid volume in chamber A that remains in the chamber after the rod is fully retracted. The default value is `1e-4` m^3.

Dead volume B

Fluid volume in chamber B that remains in the chamber after the rod is fully extended. The default value is `1e-4` m^3.

Specific heat ratio

Gas-specific heat ratio for the Hydraulic Piston Chamber blocks. The default value is `1.4`.

Cylinder orientation

Specifies cylinder orientation with respect to the globally assigned positive direction. The cylinder can be installed in two different ways, depending upon whether it exerts force in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port A exerts force in negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Hard Stop Properties Tab

Contact stiffness

Specifies the elastic property of colliding bodies for the Translational Hard Stop block. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is `1e6` N/m.

Contact damping

Specifies dissipating property of colliding bodies for the Translational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N*s/m.

Hard stop model

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` (default) — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.
- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

Transition region

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero. The default value is `1 mm`..

Initial Conditions Tab

Piston initial distance from cap A

The distance that the piston is extended at the beginning of simulation. You can set the piston position to any point within its stroke. The default value is `0`, which corresponds to the fully retracted position.

Chamber A initial pressure

Pressure in the cylinder chamber A at the beginning of simulation. The default value is `0`.

Chamber B initial pressure

Pressure in the cylinder chamber B at the beginning of simulation. The default value is `0`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Cylinder orientation**

All other block parameters are available for modification.

Global Parameters

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the cylinder chamber A.

`B`

Hydraulic conserving port associated with the cylinder chamber B.

`R`

Mechanical translational conserving port associated with the cylinder rod.

`C`

Mechanical translational conserving port associated with the cylinder clamping structure.

Examples

The Hydraulic Cylinder with Mount example illustrates simulation of a cylinder whose clamping is too flexible to be neglected. The structure compliance is represented with a spring and a damper, installed between the cylinder case and reference point. The cylinder performs forward and return strokes, and is loaded with inertia, viscous friction, and constant opposing load of 400 N.

The Custom Hydraulic Cylinder example demonstrates the use of a 4-way valve in combination with a double-acting cylinder in a simple closed-loop actuator. The example shows how to connect the blocks and set the initial orifice openings for the 4-way valve to model the forward and return strokes of the cylinder under load.

See Also

Ideal Translational Motion Sensor | Single-Acting Hydraulic Cylinder | Translational Hard Stop | Translational Hydro-Mechanical Converter

Introduced in R2006a

Double-Acting Hydraulic Cylinder (Simple)

Basic functionality of double-acting hydraulic cylinder

Library

Hydraulic Cylinders

Description

The Double-Acting Hydraulic Cylinder (Simple) block represents a simplified version of a double-acting hydraulic cylinder, developed for applications where only the basic cylinder functionality must be reproduced, in exchange for better numerical efficiency. For these reasons, such factors as fluid compressibility, friction, and leakages are assumed to be negligible. The hard stops are assumed to be fully inelastic, to eliminate any possible oscillations at the end of the stroke. The model is especially suitable for real-time and HIL (hardware-in-the-loop) simulation, if such simplifications are acceptable.

The model is described with the following equations:

$$F = A_A \cdot p_A - A_B \cdot p_B - F_c$$

$$q_A = A_A \cdot v$$

$$q_B = A_B \cdot v$$

$$\frac{dx}{dt} = v$$

$$v = v_R - v_C$$

$$F_c = \begin{cases} (x - x_E) \cdot K_p \cdot v & \text{if } x > x_E, v > 0 \\ (x - x_R) \cdot K_p \cdot v & \text{if } x < x_R, v < 0 \\ 0 & \text{otherwise} \end{cases}$$

$$x_E = S - x_0$$

$$x_R = -x_0$$

where

F	Force developed by the cylinder
v	Cylinder rod velocity
v_R, v_C	Absolute velocities of cylinder rod and cylinder case, respectively
A_A	Piston area at port A side
A_B	Piston area at port B side
p_A	Pressure at the cylinder port A
p_B	Pressure at the cylinder port B
q_A	Flow rate through port A into the cylinder
q_B	Flow rate through port B from the cylinder
x	Piston position
x_0	Initial distance between piston and cap A
F_c	Hard stop force
x_E	Distance the piston can travel to fully extend from initial position
x_R	Distance the piston can travel to fully retract from initial position
K_p	Penetration coefficient
S	Piston stroke

The main difference between this block and the Double-Acting Hydraulic Cylinder block is the representation of the hard stop:

- The Double-Acting Hydraulic Cylinder block utilizes the elastic model of the stop, which is essentially a combination of a spring and a damper, connected in parallel. The stiffness of the spring, which represents contact stiffness of colliding bodies, is very

high, resulting in high-frequency, low-magnitude oscillations at the moment the bodies collide. These oscillations can hardly be noticed experimentally, but they are difficult to process by a numerical simulation and generally decrease numerical efficiency.

- The hard stop in the Double-Acting Hydraulic Cylinder (Simple) block is represented with the inelastic model, which is essentially a viscous damper with the penetration-dependent damping coefficient. This coefficient is referred to as the penetration coefficient. With inelastic model, no oscillation is generated during an impact, resulting in improved numerical robustness and efficiency. But inelastic stop model has a feature that you must consider while selecting the model: colliding bodies continue slowly moving into each other as long as the contact is loaded with the compressing force. In real life, this phenomenon is similar to collision of two bodies separated by a sizable layer of viscous liquid. It takes some time to squeeze the liquid before bodies themselves come into contact.

Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connections A and B are hydraulic conserving ports. Port A is connected to chamber A and port B is connected to chamber B.

You can adjust the block directionality with the **Cylinder orientation** parameter.

Basic Assumptions and Limitations

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops are assumed to be fully inelastic, as explained above.

Parameters

Piston area A

Chamber A effective piston area. The default value is `0.001` m^2.

Piston area B

Chamber B effective piston area. The default value is `5e-4` m^2.

Piston stroke

Piston maximum travel between caps. The default value is `0.1` m.

Piston initial distance from cap A

The distance between the piston and cap A at the beginning of simulation. This value cannot exceed the piston stroke. The default value is `0`.

Penetration coefficient

Specifies the penetration property of colliding bodies. The higher the value of the coefficient, the less the bodies penetrate into each other, but the sharper the increase of the impact force. To find out the value of the coefficient for a particular application, it is recommended to run the same model with different hard stop models (or with different cylinder models) until they show close results. The default value of the coefficient is `1e12` N/m/(m/s).

Cylinder orientation

Specifies cylinder orientation with respect to the globally assigned positive direction. The cylinder can be installed in two different ways, depending upon whether it exerts force in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port A exerts force in negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Cylinder orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the cylinder chamber A.

`B`

Hydraulic conserving port associated with the cylinder chamber B.

R

Mechanical translational conserving port associated with the cylinder rod.

C

Mechanical translational conserving port associated with the cylinder clamping structure.

Examples

The Closed-Loop Hydraulic Actuator Model for Real-Time Simulation example shows how to use this cylinder along with other blocks optimized for real-time and HIL simulation.

See Also

Double-Acting Hydraulic Cylinder | Single-Acting Hydraulic Cylinder | Single-Acting Hydraulic Cylinder (Simple) | Translational Hard Stop | Translational Hydro-Mechanical Converter

Introduced in R2010a

Double-Acting Rotary Actuator

Double-acting hydraulic rotary actuator

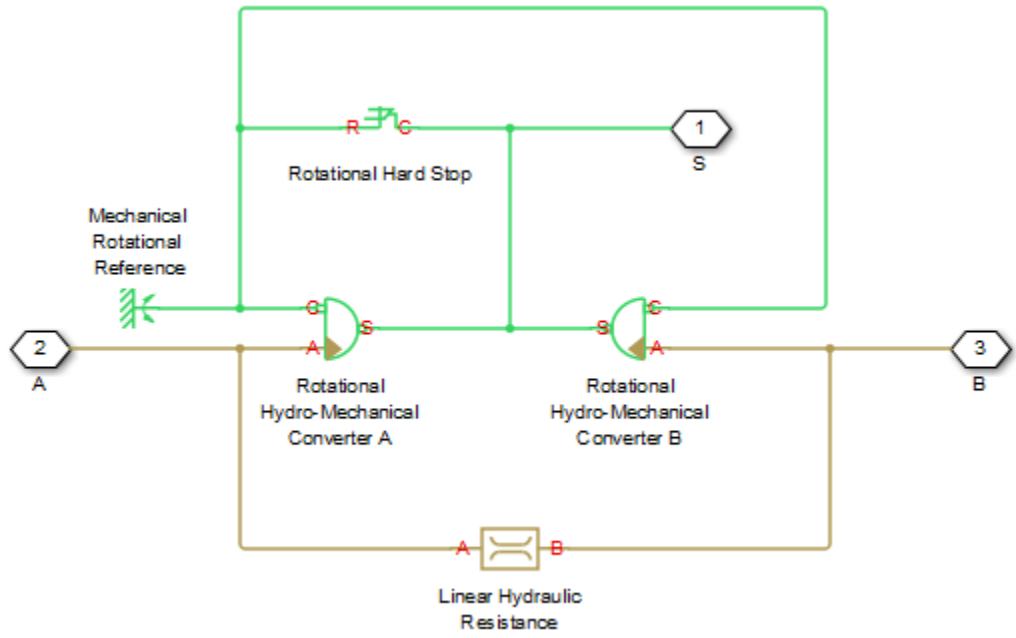
Library

Hydraulic Cylinders

Description

The Double-Acting Rotary Actuator block models a double-acting hydraulic rotary actuator, which directly converts hydraulic energy into mechanical rotational energy without employing intermediary transmissions such as rack-and-pinion, sliding spline, chain, and so on. Hydraulic fluid pumped under pressure into one of the two actuator chambers forces the shaft to rotate and generate torque. Double-acting actuators generate torque and motion in both directions.

The model of the actuator is built of Simscape Foundation library blocks. The schematic diagram of the model is shown below.



The blocks in the diagram perform the following functions:

Rotational Hydro-Mechanical Converter A	Converts hydraulics energy into mechanical rotational energy when fluid is pumped into actuator chamber A, while accounting for fluid compressibility.
Rotational Hydro-Mechanical Converter B	Converts hydraulics energy into mechanical rotational energy when fluid is pumped into actuator chamber B, while accounting for fluid compressibility.
Rotational Hard Stop	Imposes limits on shaft rotation.
Linear Hydraulic Resistance	Accounts for leakages.

Connections A and B are hydraulic conserving ports. Port A is connected to chamber A and port B is connected to chamber B. Connection S is a mechanical rotational conserving port associated with the actuator shaft.

The block directionality is adjustable and can be controlled with the **Actuator orientation** parameter.

Basic Assumptions and Limitations

No loading, such as inertia, friction, spring, and so on, is taken into account. If necessary, you can easily add them by connecting an appropriate building block to port S.

Parameters

Actuator displacement

Effective displacement of the actuator. The default value is `4.5e-5` m^3/rad.

Shaft stroke

Shaft maximum travel between stops. The default value is `5.1` rad.

Shaft initial angle

The position of the shaft at the beginning of simulation. You can set the shaft position to any angle within its stroke. The default value is `0`, which corresponds to the shaft position at the very beginning of the stroke.

Dead volume A

Fluid volume in chamber A that remains in the chamber when the shaft is positioned at the very beginning of the stroke. The default value is `1e-4` m^3.

Dead volume B

Fluid volume in chamber B that remains in the chamber when the shaft is positioned at the end of the stroke. The default value is `1e-4` m^3.

Leak coefficient

Leak coefficient for the Linear Hydraulic Resistance block. The default value is `1e-14` (m^3/s)/Pa.

Specific heat ratio

Gas-specific heat ratio for the Hydraulic Piston Chamber block. The default value is `1.4`.

Contact stiffness

Specifies the elastic property of colliding bodies for the Rotational Hard Stop block. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is `1e6` N*m/rad.

Contact damping

Specifies dissipating property of colliding bodies for the Rotational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N*m/(rad/s).

Hard stop model

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` (default) — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.
- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

Transition region

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero. The default value is `1 mm`..

Actuator orientation

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it generates torque in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port A generates torque in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Actuator orientation**

All other block parameters are available for modification.

Global Parameters

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the actuator chamber A.

B

Hydraulic conserving port associated with the actuator chamber B.

S

Mechanical rotational conserving port associated with the actuator shaft.

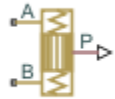
See Also

Linear Hydraulic Resistance | Rotational Hard Stop | Rotational Hydro-Mechanical Converter

Introduced in R2007a

Double-Acting Servo Cylinder

Hydraulic double-acting servo cylinder with spring-centered plunger

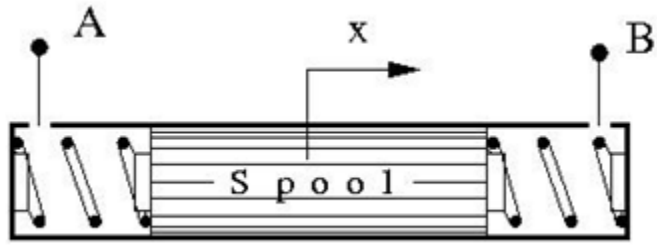


Library

Valve Actuators

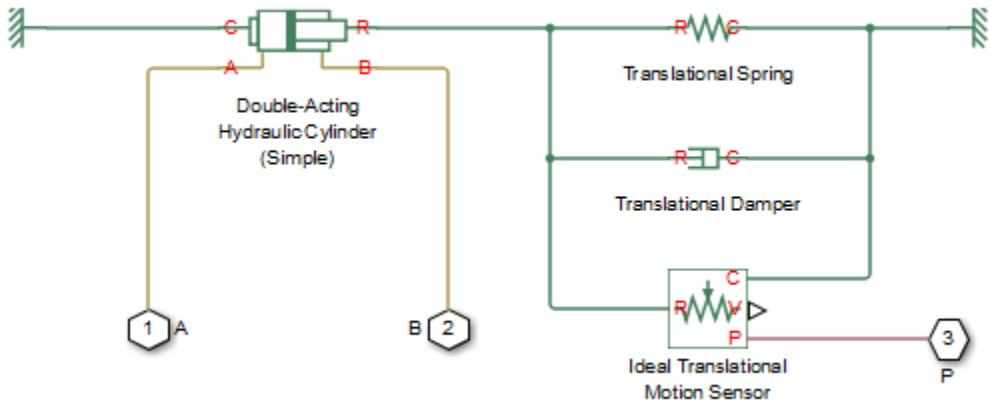
Description

The Double-Acting Servo Cylinder block models a hydraulic double-acting servo cylinder implemented as a spring-centered plunger.



Changing pressure at ports A and B shifts the spool from the neutral position in proportion to the pressure differential. Use the servo cylinder for control purposes.

The model of the cylinder uses the Double-Acting Hydraulic Cylinder (Simple), Translational Spring, Translational Damper, and Ideal Translational Motion Sensor blocks. The following figure shows the schematic diagram of the model.



The plunger is located initially in the middle of the stroke. The plunger displacement is computed with the following equations:

$$A\left(p_A - p_B\right) = k \cdot x + B \cdot v + F_{HS}$$

$$F_{HS} = \begin{cases} \left(x - S/2\right) \cdot K_p \cdot v & \text{if } x > S/2, v > 0 \\ \left(x + S/2\right) \cdot K_p \cdot abs\left(v\right) & \text{if } x < -S/2, v < 0 \\ 0 & \text{otherwise} \end{cases}$$

where

p_A, p_B	Pressures at port A and B, respectively
A	Spool face area
k	Spring rate
B	Damping coefficient
F_{HS}	Hard stop force
K_p	Stop penetration coefficient
S	Piston stroke
v	Piston velocity

x	Piston displacement

Connection P is a physical signal output port that outputs the plunger displacement from the central position. Connections A and B are hydraulic conserving ports. Pressure at port A moves the plunger in the positive direction.

Basic Assumptions and Limitations

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops are assumed to be fully inelastic. For details, see the Double-Acting Hydraulic Cylinder (Simple) block reference page.

Parameters

Piston area

The face area of the piston. The default value is `1e-4` m^2.

Piston stroke

The full piston stroke, from one hard stop to another. The piston is located initially in the middle of the stroke and can travel half a stroke in the positive and negative direction. The default value is `0.005` m.

Spring rate

The spring rate of the centering springs. The default value is `1000` N/m.

Damping coefficient

The damping coefficient in the contact between the piston and the case. The default value is `150` N/(m/s).

Stop penetration coefficient

The penetration property of the piston hard stop. The hard stop is represented as absolutely inelastic, and the property is characterized by the penetration coefficient. The default value of the coefficient is `1e12` N/m/(m/s).

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the cylinder chamber A.

B

Hydraulic conserving port associated with the cylinder chamber B.

P

Physical signal output port that outputs the piston displacement.

See Also

Double-Acting Hydraulic Cylinder (Simple) | Ideal Translational Motion Sensor | Translational Damper | Translational Spring

Introduced in R2013a

Elbow

Hydraulic resistance in elbow

Library

Local Hydraulic Resistances

Description

The Elbow block represents an elbow as a local hydraulic resistance. The pressure loss is computed with the semi-empirical formula based on pressure loss coefficient, which is determined in accordance with the Crane Co. recommendations (see [1] on page 1-476, p. A-29). Two types of elbow are considered: smoothly curved (standard) and sharp-edged (miter). The block covers elbows in the 5–100 mm and 0–90 degrees range.

The block is based on the Local Resistance block. It computes the pressure loss coefficient and passes its value, as well as the critical Reynolds number value, to the Local Resistance block, which computes the pressure loss according to the formulas explained in the reference documentation for that block.

The pressure loss for turbulent flow regime is determined according to the following formula:

$$p = K\frac{\rho}{2A^2}q\,|\,q\,|$$

where

q	Flow rate
p	Pressure loss

K	Pressure loss coefficient
A	Elbow cross-sectional area
ρ	Fluid density

The flow regime is checked in the underlying Local Resistance block by comparing the Reynolds number to the specified critical Reynolds number value. For laminar flow regime, the formula for pressure loss computation is modified, as described in the reference documentation for the Local Resistance block.

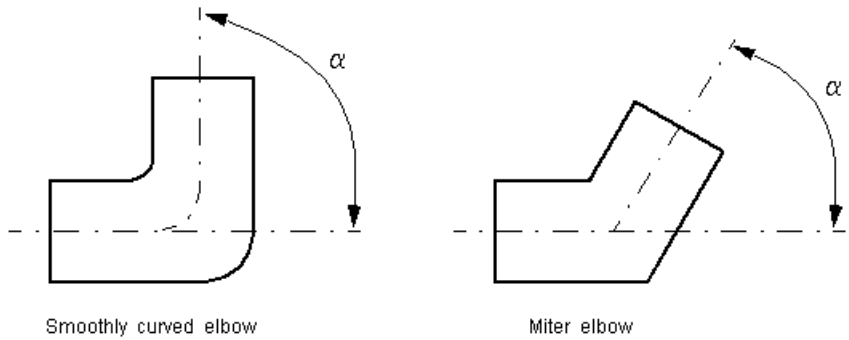
The core data for the pressure loss coefficient computation is the table-specified relationship between the friction factor f_T and the internal diameter for clean commercial steel pipes, with flow in the zone of complete turbulence (see [1] on page 1-476, p. A-26). For smoothly curved, standard 90° elbows, the pressure loss coefficient is determined with the formula

$$K = 30 f_T$$

For elbows with different angles, the coefficient is corrected with the relationship presented in [2] on page 1-476, Fig.4.6:

$$K_{corr} = \alpha(0.0142 - 3.703 \cdot 10^{-5} \alpha)$$

where α is the elbow angle in degrees ($0 \le \alpha \le 90$).



Therefore, the pressure loss coefficient for smoothly curved, standard elbows is determined with the formula

$$K_{SCE} = 30 f_T \cdot \alpha (0.0142 - 3.703 \cdot 10^{-5} \alpha)$$

For sharp-edged, miter bends the pressure loss coefficient is determined according to the table provided in [1] on page 1-476, p. A-29, as a function of the elbow diameter and angle

$$K_{ME} = f(d, \alpha)$$

where $5 \le d \le 100$ mm and $0 \le \alpha \le 90$ degrees.

Connections A and B are conserving hydraulic ports associated with the block inlet and outlet, respectively.

The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure differential is determined as

$p = p_A - p_B$.

Warning The formulas used in the Elbow block are very approximate, especially in the laminar and transient flow regions. For more accurate results, use the Local Resistance block with a table-specified *K=f(Re)* relationship.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- The elbow is assumed to be made of a clean commercial steel pipe.

Parameters

Elbow type

The parameter can have one of two values: `Smoothly curved` or `Sharp-edged (miter)`. The default value is `Smoothly curved`.

Elbow internal diameter

The internal diameter of the pipe. The value must be in the range between 5 and 100 mm. The default value is `0.01` m.

Elbow angle

The angle of the bend. The value must be in the range between 0 and 90 degrees. The default value is `90` deg.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `80`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Elbow type**
- **Laminar transition specification**

All other block parameters are available for modification.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the elbow inlet.

B

Hydraulic conserving port associated with the elbow outlet.

References

[1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M

[2] George R. Keller, *Hydraulic System Analysis*, Published by the Editors of Hydraulics & Pneumatics Magazine, 1970

See Also

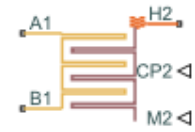
Gradual Area Change | Local Resistance | Pipe Bend | Sudden Area Change | T-junction

Introduced in R2006b

Heat Exchanger (TL)

Detailed model of a heat exchanger between a thermal liquid network and a general fluid

Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers



Description

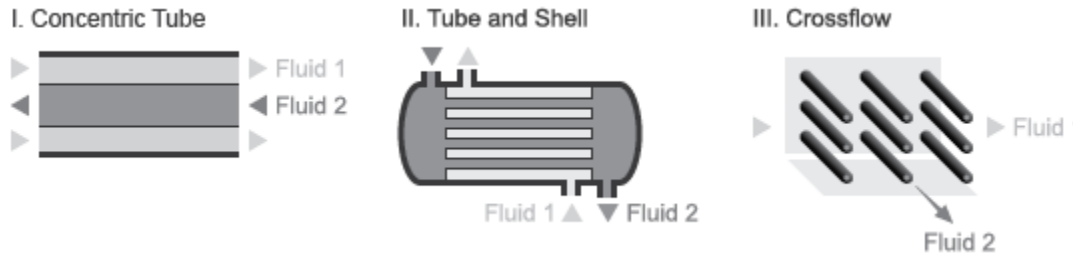
The Heat Exchanger (TL) block models the heat transfer and fluid dynamics of a heat exchanger located between a thermal liquid network and an external fluid. The fluid properties of the thermal liquid network, specified through a Thermal Liquid Settings (TL) block, do not apply to the external fluid. The fluids do not mix nor do they change phase.

The equations underpinning the model, and therefore the available parameterizations and their required inputs, depend on the active block variant. To see the active variant or to change it, right-click the block and select **Simscape > Block Choices**. There are two variants with differing degrees of assumed knowledge about the component properties:

- `Simple Model` — Provide tabulated data relating the mass flow rate at the inlets to the pressure drop between them and to the specific dissipation factor of the heat exchanger (a measure of its effectiveness). The heat transfer rate is determined from the value of the specific dissipation factor at the simulated operating conditions.
- `E-NTU Model` — Provide detailed data on the component geometry, flow configuration, and performance properties such as the thermal resistance, pressure loss coefficient, and fouling factor. The heat transfer rate is determined in part from a calculated parameter known as the *number of transfer units*, or *NTU*.

Flow Configuration

The block dialog box provides a choice of common heat exchanger configurations. These include concentric-pipe with parallel and counter flows, shell-and-tube with one or more shell passes, and cross-flow with mixed and unmixed flows. A generic configuration lets you model other heat exchangers based on tabular effectiveness data.

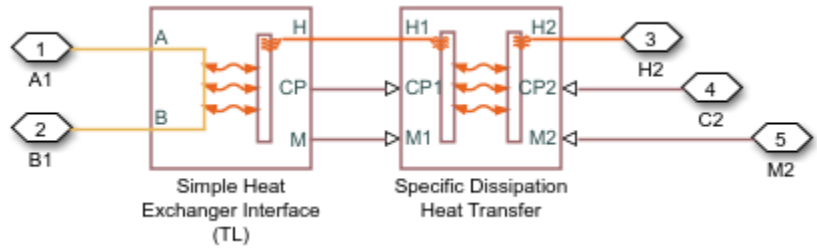


Heat Exchanger Configurations

Component Structure

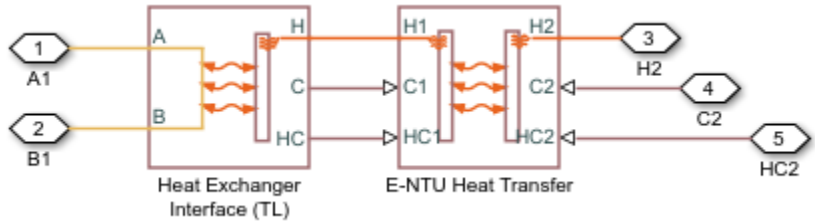
This block is a composite component assembled from simpler components found in the **Fundamental Components** library. The structural block diagram depends on the active block variant. The figures show the blocks comprising each variant and their respective connections.

In the `Simple Model` variant, a Specific Dissipation Heat Transfer block captures the heat transfer between the thermal liquid networks while a Simple Heat Exchanger Interface (TL) block captures the pressure drop and temperature change between the inlets.



Block Diagram for `Simple Model` Variant

In the `E-NTU Model` variant, an E-NTU Heat Transfer block captures the heat transfer between the thermal liquid networks while a Heat Exchanger Interface (TL) block captures the pressure drop and temperature change between the inlets.



Block Diagram for `E-NTU Model` Variant

Ports

Input

`CP2` — Isobaric specific heat of the controlled fluid

physical signal

Isobaric specific heat of the controlled fluid.

Dependencies

This port is exposed when the block variant is set to `Simple Model`. Right-click the block and select **Simscape** > **Block Choices** to change the active block variant.

`M2` — Entrance mass flow rate of the controlled fluid

physical signal

Entrance mass flow rate of the controlled fluid. Positive values indicate flow into the heat exchanger. Negative values indicate flow out of the heat exchanger.

Dependencies

This port is exposed when the block variant is set to `Simple Model`. Right-click the block and select **Simscape** > **Block Choices** to change the active block variant.

`C2` — Thermal capacity rate of external fluid

physical signal

Thermal capacity rate of the external fluid.

Dependencies

This port is exposed when the block variant is set to `E-NTU Model`. Right-click the block and select **Simscape** > **Block Choices** to change the active block variant.

`HC2` — Heat transfer coefficient for external fluid

physical signal

Fluid-wall heat transfer coefficient for the external fluid.

Dependencies

This port is exposed when the block variant is set to `E-NTU Model`. Right-click the block and select **Simscape** > **Block Choices** to change the active block variant.

Conserving

`A1` — Thermal liquid inlet

thermal liquid

Opening through which thermal liquid can enter and exit the heat exchanger.

`B1` — Thermal liquid inlet

thermal liquid

Opening through which thermal liquid can enter and exit the heat exchanger.

`H2` — Inlet temperature of external fluid

thermal

Thermal port associated with the inlet temperature of the external fluid.

Parameters

Block Variant: `Simple Model`

Heat Transfer Tab

`Thermal Liquid mass flow rate vector, mdot1` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, .6, .7, 1, 1.4, 1.9, 2.3] kg/s` (default) | *M*-element array with units of mass/time

Array of mass flow rates at the thermal liquid inlet. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Controlled fluid mass flow rate vector, mdot2` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, 1, 1.3, 1.7, 2, 2.6, 3.3] kg/s` (default) | *N*-element array with units of mass/time

Array of mass flow rates at the inlet for the controlled fluid. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Specific dissipation table, SD(mdot1, mdot2)` — Specific dissipation values corresponding to specified mass flow rates

8-by-8 matrix with units of `kW/K` (default) | *M*-by-*N* matrix with units of power/temperature

Matrix of specific dissipation values corresponding to the specified mass flow rate arrays for the thermal liquids and the controlled fluid. The block uses the tabulated data to calculate the heat transfer at the simulated operating conditions.

`Check if violating maximum specific dissipation` — Warning condition for out-of-bounds specific dissipation

`Warning` (default) | `None`

Option to warn if the specific dissipation falls outside the bounds of the specified tabulated data.

Pressure Loss Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data
`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates
`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet corresponding to the tabulated mass flow rate data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified
`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified
`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-3 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Thermal Liquid volume` — Volume of thermal liquid inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of thermal liquid occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A1 and B1` — Flow area at the thermal liquid inlets
`0.01 m^2` (default) | scalar with units of area

Flow area at the thermal liquid inlets. Inlets **A1** and **B1** are assumed to be identical in size.

Block Variant: `E-NTU Model`

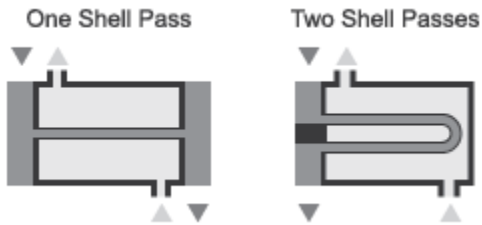
Common Tab

`Flow arrangement` — Geometry of flow through heat exchanger
`Parallel or counter flow` (default) | `Shell and tube` | `Cross flow` | `Generic - effectiveness table`

Geometry of flow through the heat exchanger. Select `Generic – effectiveness table` to provide effectiveness data for any flow arrangements not explicitly provided.

`Number of shell passes` — Geometry of shell-and-tube configuration
`1` (default) | positive unitless scalar

Number of passes that the heat exchanger tube makes along the length of the shell between the fluid inlet and outlet. Increasing the number of shell passes increases the area of heat transfer between the fluids.



One- and Two-Pass Configurations

`Cross flow type` — Mixing conditions of the heat exchanger fluids
`Both fluids mixed` (default) | `Both fluids unmixed` | `Thermal Liquid mixed & Controlled Fluid unmixed` | `Thermal Liquid unmixed & Controlled Fluid mixed`

Mixing condition of each fluid. Mixing impacts the heat transfer correlations used during simulation. Each fluid mixes only with itself—in its compartment in isolation from the second fluid.

`Number of heat transfer units vector, NTU` — Values of the number of heat transfer units at which to specify the effectiveness data
`[.5, 1, 2, 3, 4, 5]` (default) | *M*-element unitless array

Values of the number of heat transfer units (*NTU*) at which to specify the tabulated effectiveness data. Each value corresponds to a row in the effectiveness lookup table. The number of transfer units is defined as:

$$NTU = \frac{A_s U}{C_{min}},$$

where:

- A_S is the heat transfer surface area.
- U is the overall heat transfer coefficient.
- C_{min} is the smallest of the thermal capacity rates of the two fluids.

`Thermal capacity ratio vector, CR` — Values of the thermal capacity ratio at which to specify the effectiveness data
`[0, .25, .5, .75, 1]` (default) | *N*-element unitless array

Values of the thermal capacity ratio at which to specify the tabulated effectiveness data. Each value corresponds to a column in the effectiveness lookup table. The thermal capacity ratio is defined as:

$$C_r = \frac{C_{min}}{C_{max}},$$

where C_{min} is the smallest of the thermal capacity rates of the two fluids and C_{max} is the largest.

`Effectiveness table, E(NTU,CR)` — Effectiveness data at the specified values of the number of heat transfer units and thermal capacity ratio
6-by-5 unitless matrix (default) | *M*-by-*N* unitless matrix

Matrix with the heat exchange effectiveness values corresponding to the specified values of the number of heat transfer units and capacity ratio. The matrix rows correspond to

different values of the number of heat transfer units. The matrix columns correspond to different values of the thermal capacity ratio.

`Wall thermal resistance` — Thermal resistance of the wall between the two fluids
`1.6e-4 K/W` (default) | scalar with units of temperature/power

Thermal resistance of the wall between the two fluids.

Thermal Liquid Tab

`Minimum free-flow area` — Flow area based on the smallest tube spacing or corrugation pitch
`0.01 m^2` (default) | positive scalar with units of area

Aggregate flow area free of obstacles based on the smallest tube spacing or corrugation pitch.

`Hydraulic diameter for pressure loss` — Ratio of flow area to channel perimeter
`0.1 m` (default) | positive scalar with units of length

Hydraulic diameter of the tubes or channels comprising the heat exchange interface. The hydraulic diameter is the ratio of the flow cross-sectional area to the channel perimeter.

`Thermal Liquid volume` — Volume of fluid inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of thermal liquid occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Laminar flow upper Reynolds number limit` — Reynolds number below which the flow is laminar
`2000` (default) | positive unitless scalar

Reynolds number below which the flow is laminar. The flow transitions to turbulent above this number and it becomes fully turbulent at the number specified in the **Turbulent flow lower Reynolds number limit** parameter.

`Turbulent flow lower Reynolds number limit` — Reynolds number above which the flow is turbulent
`4000` (default) | positive unitless scalar

Reynolds number above which the flow is turbulent. The flow transitions to laminar below this number and it becomes fully laminar at the number specified in the **Laminar flow upper Reynolds number limit** parameter.

`Pressure loss parameterization` — Parameterization of the pressure-loss calculation

`constant loss coefficient` (default) | `correlations for tubes` | `Tabulated data - Darcy friction factor vs. Reynolds number` | `Tabulated data - Euler number vs. Reynolds number`

Parameterization of the pressure-loss calculation. The default parameterization, `constant loss coefficient`, provides the simplest option. Select a different parameterization if you have detailed data on the Darcy friction factor or Euler number.

`Pressure loss coefficient` — Value of the pressure loss coefficient from inlet to outlet

`0.1` (default) | unitless scalar between `0` and `1`

Dimensionless number used to compute the pressure loss between the inlet and outlet. The pressure loss coefficient is assumed constant and the same for direct and reverse flows.

`Length of flow path from inlet to outlet` — Distance traversed by the fluid from inlet to outlet

`1 m` (default) | unitless scalar with units of length

Distance traversed by the fluid from inlet to outlet.

`Aggregate equivalent length of local resistances` — Pressure loss due to resistances expressed as a length

`0.1 m` (default) | unitless scalar with units of length

Pressure loss due to local resistances such as bends, inlets, and fittings, expressed as the equivalent length of those resistances.

`Internal surface absolute roughness` — Average height of internal surface defects

`15e-6 m` (default) | unitless scalar with units of length

Average height of all surface defects on the internal surface of the pipe. The surface roughness enables the calculation of the friction factor in the turbulent flow regime.

`Shape factor for laminar flow viscous friction` — Proportionality between convective and conductive heat transfer in the laminar regime
`64` (default) | positive unitless scalar

Proportionality constant between convective and conductive heat transfer in the laminar regime. The shape factor encodes the effects of component geometry on the laminar friction losses.

`Reynolds number vector for Darcy friction factor` — Values of the Reynolds number at which to specify the Darcy friction factor data
`12-element unitless array` (default) | *K*-element unitless array

Values of the Reynolds number at which to specify the tabulated Darcy friction factor data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to an element in that table.

`Darcy friction factor vector` — Darcy friction factor data corresponding to the specified values of the Reynolds number
`12-element unitless array` (default) | *K*-element unitless array

Tabulated Darcy friction factor data corresponding to the specified values of the Reynolds number. The block uses this vector to create a lookup table for the Darcy friction factor.

`Reynolds number vector for Euler number` — Values of the Reynolds number at which to specify the Euler number data
`[50, 500, 1000, 2000]` (default) | *J*-element unitless array

Values of the Reynolds number at which to specify the tabulated euler number data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to an element in that table.

`Euler number vector` — Euler number corresponding to the specified values of the Reynolds number
`[4.4505, .6864, .4791, .3755]` (default) | *J*-element unitless array

Tabulated Euler number data corresponding to the specified values of the Reynolds number. The block uses this vector to create a lookup table for the Euler number.

`Heat transfer parameterization` — Parameterization of the heat transfer calculation
(default) | | `Constant heat transfer coefficient` | `Correlations for tubes` | `Tabulated data - Colburn factor vs. Reynolds number` | `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`

Parameterization used to calculate the heat transfer rate between the heat exchanger fluids.

`Heat transfer surface area` — Area available for heat transfer between the heat exchanger fluids
`0.4 m^2` (default) | scalar with units of area

Aggregate surface area available for heat transfer between the heat exchanger fluids.

`Length of flow path for heat transfer` — Distance along which heat transfer takes place
`1 m` (default) |

Distance along which heat transfer takes place.

`Nusselt number for laminar flow heat transfer` — Proportionality between convective and conductive heat transfer in the laminar regime
`3.66` (default) | positive unitless scalar

Proportionality constant between convective and conductive heat transfer in the laminar regime. This parameter enables the calculation of convective heat transfer rates in laminar flows. The appropriate value to use depends on component geometry.

`Reynolds number vector for Colburn factor` — Values of the Reynolds number at which to specify the Colburn factor data
`[100, 150, 1000]` (default) | *L*-element unitless array

Values of the Reynolds number at which to specify the tabulated Colburn factor data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to an element in that table.

`Colburn factor vector` — Colburn factor data corresponding to the specified values of the Reynolds number
`[.019, .013, .002]` (default) | *L*-element unitless array

Tabulated Colburn factor data corresponding to the specified values of the Reynolds number. The block uses this vector to create a lookup table for the Colburn factor.

`Reynolds number vector for Nusselt number` — Values of the Reynolds number at which to specify the Nusselt number data
`[100, 150, 1000]` (default) | *L*-element unitless array

Values of the Reynolds number at which to specify the tabulated Nusselt number data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to a row in that table.

`Prandtl number vector for Nusselt number` — Values of the Prandtl number at which to specify the Nusselt number data
`[1, 10]` (default) | *P*-element unitless array

Values of the Reynolds number at which to specify the tabulated Nusselt number data. The block uses this data to construct a lookup table. Each Prandtl number corresponds to a column in that table.

`Nusselt number table, Nu (Re,Pr)` — Nusselt number data corresponding to the specified values of the Reynolds and Prandlt numbers
`[3.72, 4.21; 3.75, 4.44; 4.21, 7.15]` (default) | *L*-by-*P* unitless matrix

Tabulated Nusselt number data corresponding to the specified values of the Reynolds and Prandlt numbers. The block uses this vector to create a lookup table for the Nusselt number, with rows corresponding to different Reynolds numbers and columns corresponding to different Prandtl numbers.

`Liquid-wall heat transfer coefficient` — Heat transfer coefficient at the liquid-wall interface
`100 W/(m^2 * K)` (default) | | scalar with units of power/area/temperature

Heat transfer coefficient between the thermal liquid and the heat-transfer interface.

`Fouling factor` — Thermal resistance due to dirt deposits on the heat transfer interface
`1.6e-4 K/W` (default) | scalar with units of temperature/power

Empirical parameter used to quantify the increased thermal resistance due to dirt deposits on the heat transfer interface.

Controlled Fluid Tab

`Heat transfer surface area` — Area of the interface between the thermal liquid and the controlled fluid
`0.4 m^2` (default)

Aggregate area through which heat will flow when driven by a temperature gradient between the thermal liquid and the controlled fluid.

`Fouling factor` — Measure of the thermal resistance on the controlled-fluid side of the thermal interface
`1e-4 m^2*K/W` (default) | scalar with units of (temperature*area)/power

Empirical measure of the resistance to heat flow—typically the result of dirt deposits—on the controlled-fluid side of the thermal interface.

`Minimum fluid-wall heat transfer coefficient` — Measure of the thermal resistance on the controlled-fluid side of the thermal interface
`5 W/(m^2 * K)` (default) | scalar with units of power/(temperature*area)

Lowest value that the heat transfer coefficient can take on the controlled-fluid side of the thermal interface. The coefficient, specified through physical signal port HC2, saturates at this value.

Effects and Initial Conditions

`Thermal Liquid dynamic compressibility` — Option to model pressure dynamics inside the heat exchanger
`On` (default) | `Off`

Option to model the pressure dynamics inside the heat exchanger. Setting this parameter to `Off` removes the pressure derivative terms from the component energy and mass conservation equations. The pressure inside the heat exchanger is then reduced to the weighted average of the two port pressures.

`Thermal Liquid initial temperature` — Temperature inside the heat exchanger at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of thermal liquid at the start of simulation.

`Thermal Liquid initial pressure` — Pressure inside the heat exchanger at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of thermal liquid at the start of simulation.

See Also

E-NTU Heat Transfer

Introduced in R2016a

Heat Exchanger (TL-TL)

Detailed model of a heat exchanger between two thermal liquid networks

Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers

Description

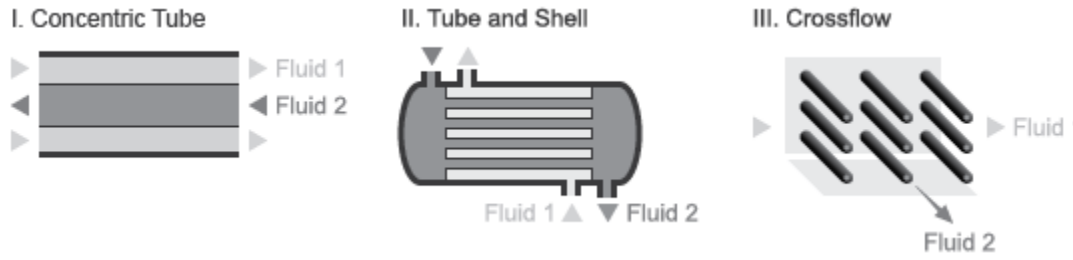
The Heat Exchanger (TL-TL) block models the heat transfer and fluid dynamics of a heat exchanger located between two thermal liquid networks. Each network must connect to a separate Thermal Liquid Settings (TL) block. The fluids do not mix nor do they change phase.

The equations underpinning the model, and therefore the available parameterizations and their required inputs, depend on the active block variant. To see the active variant or to change it, right-click the block and select **Simscape > Block Choices**. There are two variants with differing degrees of assumed knowledge about the component properties:

- `Simple Model` — Provide tabulated data relating the mass flow rate at the inlets to the pressure drop between them and to the specific dissipation factor of the heat exchanger (a measure of its effectiveness). The heat transfer rate is determined from the value of the specific dissipation factor at the simulated operating conditions.
- `E-NTU Model` — Provide detailed data on the component geometry, flow configuration, and performance properties such as the thermal resistance, pressure loss coefficient, and fouling factor. The heat transfer rate is determined in part from a calculated parameter known as the *number of transfer units*, or *NTU*.

Flow Configuration

The block dialog box provides a choice of common heat exchanger configurations. These include concentric-pipe with parallel and counter flows, shell-and-tube with one or more shell passes, and cross-flow with mixed and unmixed flows. A generic configuration lets you model other heat exchangers based on tabular effectiveness data.

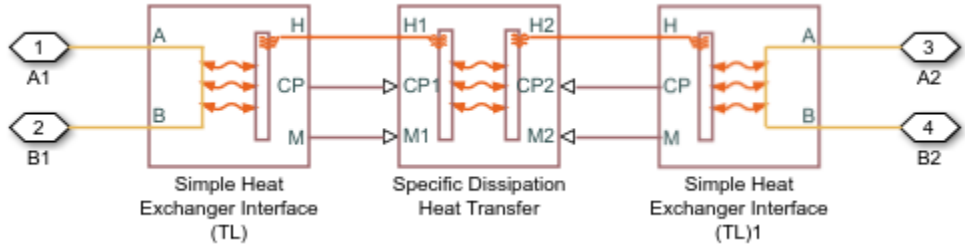


Heat Exchanger Configurations

Component Structure

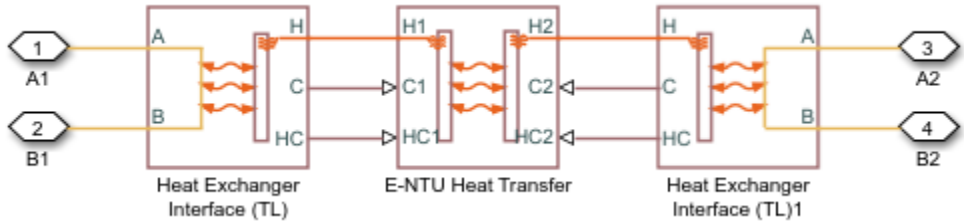
This block is a composite component assembled from simpler components found in the **Fundamental Components** library. The structural block diagram depends on the active block variant. The figures show the blocks comprising each variant and their respective connections.

In the `Simple Model` variant, a Specific Dissipation Heat Transfer block captures the heat transfer between the thermal liquid networks while two Simple Heat Exchanger Interface (TL) blocks capture the pressure drop and temperature change between the inlets.



Block Diagram for `Simple Model` Variant

In the `E-NTU Model` variant, an E-NTU Heat Transfer block captures the heat transfer between the thermal liquid networks while two Heat Exchanger Interface (TL) blocks capture the pressure drop and temperature change between the inlets.



Block Diagram for `E-NTU Model` Variant

Ports

Conserving

`A1` — Thermal liquid 1 inlet
thermal liquid

Opening through which thermal liquid 1 can enter and exit the heat exchanger.

`B1` — Thermal liquid 1 inlet
thermal liquid

Opening through which thermal liquid 1 can enter and exit the heat exchanger.

`A2` — Thermal liquid 2 inlet
thermal liquid

Opening through which thermal liquid 2 can enter and exit the heat exchanger.

`B2` — Thermal liquid 2 inlet
thermal liquid

Opening through which thermal liquid 2 can enter and exit the heat exchanger.

Parameters

Block Variant: `Simple Model`

Heat Transfer Tab

`Thermal Liquid 1 mass flow rate vector, mdot1` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, .6, .7, 1, 1.4, 1.9, 2.3] kg/s` (default) | *M*-element array with units of mass/time

Array of mass flow rates at the inlet for thermal liquid 1. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Thermal Liquid 2 mass flow rate vector, mdot2` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, 1, 1.3, 1.7, 2, 2.6, 3.3] kg/s` (default) | *N*-element array with units of mass/time

Array of mass flow rates at the inlet for thermal liquid 2. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Specific dissipation table, SD(mdot1, mdot2)` — Specific dissipation values corresponding to specified mass flow rates

8-by-8 matrix with units of `kW/K` (default) | *M*-by-*N* matrix with units of power/temperature

Matrix of specific dissipation values corresponding to the specified mass flow rate arrays for thermal liquids 1 and 2. The block uses the tabulated data to calculate the heat transfer at the simulated operating conditions.

`Check if violating maximum specific dissipation` — Warning condition for out-of-bounds specific dissipation

`Warning` (default) | `None`

Option to warn if the specific dissipation falls outside the bounds of the specified tabulated data.

Thermal Liquid 1|2 Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data
`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates for thermal liquid 1|2 at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates
`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet for thermal liquid 1|2 corresponding to the specified mass flow rate tabulated data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified
`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified
`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-3 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Thermal Liquid 1|2 volume` — Volume of fluid 1|2 inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of thermal liquid 1|2 occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A1|A2 and B1|B2` — Flow area at the thermal liquid inlets
`0.01 m^2` (default) | scalar with units of area

Flow area at the thermal liquid inlets. Inlets **A** and **B** are assumed to be identical in size for each fluid.

Block Variant: `E-NTU Model`

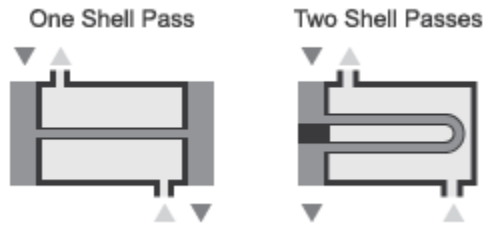
Common Tab

`Flow arrangement` — Geometry of flow through heat exchanger
`Parallel or counter flow` (default) | `Shell and tube` | `Cross flow` | `Generic - effectiveness table`

Geometry of flow through the heat exchanger. Select `Generic – effectiveness table` to provide effectiveness data for any flow arrangements not explicitly provided.

`Number of shell passes` — Geometry of shell-and-tube configuration
1 (default) | positive unitless scalar

Number of passes that the heat exchanger tube makes along the length of the shell between the fluid inlet and outlet. Increasing the number of shell passes increases the area of heat transfer between the fluids.



One- and Two-Pass Configurations

`Cross flow type` — Mixing conditions of the heat exchanger fluids
`Both fluids mixed` (default) | `Both fluids unmixed` | `Thermal Liquid 1 mixed & Thermal Liquid 2 unmixed` | `Thermal Liquid 1 unmixed & Thermal Liquid 2 mixed`

Mixing condition of each fluid. Mixing impacts the heat transfer correlations used during simulation. Each fluid mixes only with itself—in its compartment in isolation from the second fluid.

`Number of heat transfer units vector, NTU` — Values of the number of heat transfer units at which to specify the effectiveness data
`[.5, 1, 2, 3, 4, 5]` (default) | *M*-element unitless array

Values of the number of heat transfer units (*NTU*) at which to specify the tabulated effectiveness data. Each value corresponds to a row in the effectiveness lookup table. The number of transfer units is defined as:

$$NTU = \frac{A_s U}{C_{min}},$$

where:

- A_S is the heat transfer surface area.
- U is the overall heat transfer coefficient.
- C_{min} is the smallest of the thermal capacity rates of the two fluids.

`Thermal capacity ratio vector, CR` — Values of the thermal capacity ratio at which to specify the effectiveness data
`[0, .25, .5, .75, 1]` (default) | *N*-element unitless array

Values of the thermal capacity ratio at which to specify the tabulated effectiveness data. Each value corresponds to a column in the effectiveness lookup table. The thermal capacity ratio is defined as:

$$C_r = \frac{C_{min}}{C_{max}},$$

where C_{min} is the smallest of the thermal capacity rates of the two fluids and C_{max} is the largest.

`Effectiveness table, E(NTU,CR)` — Effectiveness data at the specified values of the number of heat transfer units and thermal capacity ratio
6-by-5 unitless matrix (default) | *M*-by-*N* unitless matrix

Matrix with the heat exchange effectiveness values corresponding to the specified values of the number of heat transfer units and capacity ratio. The matrix rows correspond to

different values of the number of heat transfer units. The matrix columns correspond to different values of the thermal capacity ratio.

`Wall thermal resistance` — Thermal resistance of the wall between the two fluids
`1.6e-4 K/W` (default) | scalar with units of temperature/power

Thermal resistance of the wall between the two fluids.

Thermal Liquid 1|2 Tab

`Minimum free-flow area` — Flow area based on the smallest tube spacing or corrugation pitch
`0.01 m^2` (default) | positive scalar with units of area

Aggregate flow area free of obstacles based on the smallest tube spacing or corrugation pitch.

`Hydraulic diameter for pressure loss` — Ratio of flow area to channel perimeter
`0.1 m` (default) | positive scalar with units of length

Hydraulic diameter of the tubes or channels comprising the heat exchange interface. The hydraulic diameter is the ratio of the flow cross-sectional area to the channel perimeter.

`Thermal Liquid 1|2 volume` — Volume of fluid 1|2 inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of thermal liquid 1|2 occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Laminar flow upper Reynolds number limit` — Reynolds number below which the flow is laminar
`2000` (default) | positive unitless scalar

Reynolds number below which the flow is laminar. The flow transitions to turbulent above this number and it becomes fully turbulent at the number specified in the **Turbulent flow lower Reynolds number limit** parameter.

`Turbulent flow lower Reynolds number limit` — Reynolds number above which the flow is turbulent
`4000` (default) | positive unitless scalar

Reynolds number above which the flow is turbulent. The flow transitions to laminar below this number and it becomes fully laminar at the number specified in the **Laminar flow upper Reynolds number limit** parameter.

`Pressure loss parameterization` — Parameterization of the pressure-loss calculation

`constant loss coefficient` (default) | `correlations for tubes` | `Tabulated data - Darcy friction factor vs. Reynolds number` | `Tabulated data - Euler number vs. Reynolds number`

Parameterization of the pressure-loss calculation. The default parameterization, `constant loss coefficient`, provides the simplest option. Select a different parameterization if you have detailed data on the Darcy friction factor or Euler number.

`Pressure loss coefficient` — Value of the pressure loss coefficient from inlet to outlet

`0.1` (default) | unitless scalar between `0` and `1`

Dimensionless number used to compute the pressure loss between the inlet and outlet. The pressure loss coefficient is assumed constant and the same for direct and reverse flows.

`Length of flow path from inlet to outlet` — Distance traversed by the fluid from inlet to outlet

`1 m` (default) | unitless scalar with units of length

Distance traversed by the fluid from inlet to outlet.

`Aggregate equivalent length of local resistances` — Pressure loss due to resistances expressed as a length

`0.1 m` (default) | unitless scalar with units of length

Pressure loss due to local resistances such as bends, inlets, and fittings, expressed as the equivalent length of those resistances.

`Internal surface absolute roughness` — Average height of internal surface defects

`15e-6 m` (default) | unitless scalar with units of length

Average height of all surface defects on the internal surface of the pipe. The surface roughness enables the calculation of the friction factor in the turbulent flow regime.

`Shape factor for laminar flow viscous friction` — Proportionality between convective and conductive heat transfer in the laminar regime
`64` (default) | positive unitless scalar

Proportionality constant between convective and conductive heat transfer in the laminar regime. The shape factor encodes the effects of component geometry on the laminar friction losses.

`Reynolds number vector for Darcy friction factor` — Values of the Reynolds number at which to specify the Darcy friction factor data
`12-element unitless array` (default) | *K*-element unitless array

Values of the Reynolds number at which to specify the tabulated Darcy friction factor data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to an element in that table.

`Darcy friction factor vector` — Darcy friction factor data corresponding to the specified values of the Reynolds number
`12-element unitless array` (default) | *K*-element unitless array

Tabulated Darcy friction factor data corresponding to the specified values of the Reynolds number. The block uses this vector to create a lookup table for the Darcy friction factor.

`Reynolds number vector for Euler number` — Values of the Reynolds number at which to specify the Euler number data
`[50, 500, 1000, 2000]` (default) | *J*-element unitless array

Values of the Reynolds number at which to specify the tabulated euler number data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to an element in that table.

`Euler number vector` — Euler number corresponding to the specified values of the Reynolds number
`[4.4505, .6864, .4791, .3755]` (default) | *J*-element unitless array

Tabulated Euler number data corresponding to the specified values of the Reynolds number. The block uses this vector to create a lookup table for the Euler number.

`Heat transfer parameterization` — Parameterization of the heat transfer calculation
`Constant heat transfer coefficient` (default) | `Correlations for tubes` | `Tabulated data - Colburn factor vs. Reynolds number` | `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`

Parameterization used to calculate the heat transfer rate between the heat exchanger fluids.

`Heat transfer surface area` — Area available for heat transfer between the heat exchanger fluids
`0.4 m^2` (default) | scalar with units of area

Aggregate surface area available for heat transfer between the heat exchanger fluids.

`Length of flow path for heat transfer` — Distance along which heat transfer takes place
`1 m` (default) | scalar with units of length

Distance along which heat transfer takes place.

`Nusselt number for laminar flow heat transfer` — Proportionality between convective and conductive heat transfer in the laminar regime
`3.66` (default) | positive unitless scalar

Proportionality constant between convective and conductive heat transfer in the laminar regime. This parameter enables the calculation of convective heat transfer rates in laminar flows. The appropriate value to use depends on component geometry.

`Reynolds number vector for Colburn factor` — Values of the Reynolds number at which to specify the Colburn factor data
`[100, 150, 1000]` (default) | *L*-element unitless array

Values of the Reynolds number at which to specify the tabulated Colburn factor data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to an element in that table.

`Colburn factor vector` — Colburn factor data corresponding to the specified values of the Reynolds number
`[.019, .013, .002]` (default) | *L*-element unitless array

Tabulated Colburn factor data corresponding to the specified values of the Reynolds number. The block uses this vector to create a lookup table for the Colburn factor.

`Reynolds number vector for Nusselt number` — Values of the Reynolds number at which to specify the Nusselt number data
`[100, 150, 1000]` (default) | *L*-element unitless array

Values of the Reynolds number at which to specify the tabulated Nusselt number data. The block uses this data to construct a lookup table. Each Reynolds number corresponds to a row in that table.

`Prandtl number vector for Nusselt number` — Values of the Prandtl number at which to specify the Nusselt number data
`[1, 10]` (default) | *P*-element unitless array

Values of the Reynolds number at which to specify the tabulated Nusselt number data. The block uses this data to construct a lookup table. Each Prandtl number corresponds to a column in that table.

`Nusselt number table, Nu (Re,Pr)` — Nusselt number data corresponding to the specified values of the Reynolds and Prandlt numbers
`[3.72, 4.21; 3.75, 4.44; 4.21, 7.15]` (default) | *L*-by-*P* unitless matrix

Tabulated Nusselt number data corresponding to the specified values of the Reynolds and Prandlt numbers. The block uses this vector to create a lookup table for the Nusselt number, with rows corresponding to different Reynolds numbers and columns corresponding to different Prandtl numbers.

`Liquid-wall heat transfer coefficient` — Heat transfer coefficient at the liquid-wall interface
`100 W/(m^2 * K)` (default) | scalar with units of power/area/temperature

Heat transfer coefficient between the thermal liquid and the heat-transfer interface.

`Fouling factor` — Thermal resistance due to dirt deposits on the heat transfer interface
`1.6e-4 K/W` (default) | scalar with units of temperature/power

Empirical parameter used to quantify the increased thermal resistance due to dirt deposits on the heat transfer interface.

Effects and Initial Conditions

`Thermal Liquid 1|2 dynamic compressibility` — Option to model pressure dynamics inside the heat exchanger
`On` (default) | `Off`

Option to model the pressure dynamics inside the heat exchanger. Setting this parameter to `Off` removes the pressure derivative terms from the component energy and mass conservation equations. The pressure inside the heat exchanger is then reduced to the weighted average of the two port pressures.

`Thermal Liquid 1|2 initial temperature` — Temperature inside the heat exchanger at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of thermal liquid 1|2 at the start of simulation.

`Thermal Liquid 1|2 initial pressure` — Pressure inside the heat exchanger at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of thermal liquid 1|2 at the start of simulation.

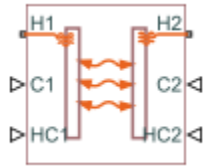
See Also

E-NTU Heat Transfer

Introduced in R2016a

E-NTU Heat Transfer

Detailed heat transfer model between two general fluids



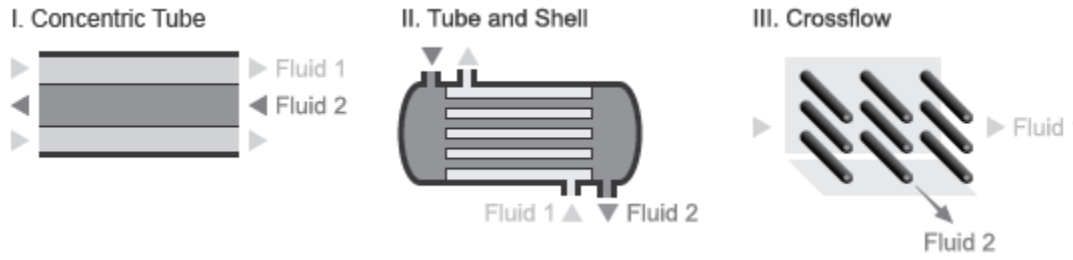
Library

Fluid Network Interfaces/Heat Exchangers/Fundamental Components

Description

The E-NTU Heat Transfer block models the heat exchange between two general fluids based on the standard Effectiveness-NTU method. The fluid thermal properties are specified explicitly through Simscape physical signals. Combine with the Heat Exchanger Interface (TL) block to model the pressure drop and temperature change between the inlet and outlet of a heat exchanger.

The block dialog box provides a choice of common heat exchanger configurations. These include concentric-pipe with parallel and counter flows, shell-and-tube with one or more shell passes, and cross-flow with mixed and unmixed flows. A generic configuration lets you model other heat exchangers based on tabular effectiveness data.



Heat Exchanger Configurations

Heat Transfer Rate

The E-NTU model defines the heat transfer rate between fluids 1 and 2 in terms of an effectiveness parameter ε:

$$Q_1 = -Q_2 = Q_{Max}, \quad 0 < \varepsilon < 1,$$

where:

- Q_1 and Q_2 are the heat transfer rates into fluid 1 and fluid 2.
- Q_{Max} is the maximum possible heat transfer rate between fluid 1 and fluid 2 at a given set of operating conditions.
- ε is the effectiveness parameter.

The maximum possible heat transfer rate between the two fluids is

$$Q_{Max} = C_{Min}\left(T_{1,In} - T_{2,In}\right),$$

where:

- C_{Min} is the minimum value of the thermal capacity rate:

  $$C_{Min} = min\left(\dot{m}_1 C_{p,1}, \dot{m}_2 C_{p,2}\right)$$

- $T_{1,In}$ and $T_{2,In}$ are the inlet temperatures of fluid 1 and fluid 2.

- $\dot{m}_1$ and $\dot{m}_2$ are the mass flow rates of fluid 1 and fluid 2 into the heat exchanger volume through the inlet.
- $c_{p,1}$ and $c_{p,2}$ are the specific heat coefficients at constant pressure of fluid 1 and fluid 2. The **Minimum fluid-wall heat transfer coefficient** parameter in the block dialog box sets a lower bound on the allowed values of the heat transfer coefficients.

Heat Exchange Effectiveness

The heat exchange effectiveness calculations depend on the flow arrangement type selected in the block dialog box. For all but `Generic — effectiveness table`, the block computes the thermal exchange effectiveness through analytical expressions written in terms of the number of transfer units (NTU) and thermal capacity ratio. The number of transfer units is defined as

$$NTU = \frac{U_{Overall} A_{Heat}}{C_{Min}} = \frac{1}{C_{Min} R_{Overall}},$$

where:

- NTU is the number of transfer units.
- $U_{Overall}$ is the overall heat transfer coefficient between fluid 1 and fluid 2.
- $R_{Overall}$ is the overall thermal resistance between fluid 1 and fluid 2.
- A_{Heat} is aggregate area of the primary and secondary, or finned, heat transfer surfaces.

The thermal capacity ratio is defined as

$$C_{rel} = \frac{C_{Min}}{C_{Max}}$$

where:

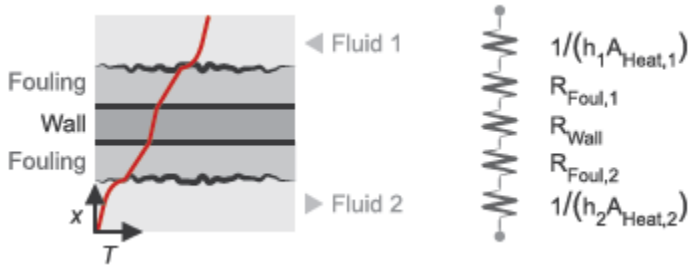
- C_{rel} is the thermal capacity ratio.

The overall heat transfer coefficient and thermal resistance used in the NTU calculation are functions of the heat transfer mechanisms at work. These mechanisms include convective heat transfer between the fluids and the heat exchanger interface and conduction through the interface wall [2 on page 1-512]:

$$R_{Overall} = \frac{1}{U_{Overall} A_{Heat}} = \frac{1}{h_1 A_{Heat,1}} + R_{Foul,1} + R_{Wall} + R_{Foul,2} + \frac{1}{h_2 A_{Heat,2}},$$

where:

- h_1 and h_2 are the heat transfer coefficients between fluid 1 and the interface wall and between fluid 2 and the interface wall.
- $A_{Heat,1}$ and $A_{Heat,2}$ are the heat transfer surface areas on the fluid-1 and fluid-2 sides.
- $R_{Foul,1}$ and $R_{Foul,2}$ are the fouling resistances on the fluid-1 and fluid-2 sides.
- R_{Wall} is the interface wall thermal resistance.



Heat Transfer From Fluid 1 to Fluid 2

The tables show some of the analytical expressions used to compute the heat exchange effectiveness [1 on page 1-512]. The parameter N refers to the number of shell passes and the parameter ε_1 to the effectiveness for a single shell pass.

Concentric Tubes	
Counter Flow	$\varepsilon = \begin{cases} \dfrac{1-\exp\left[-NTU\left(1-C_{rel}\right)\right]}{1-C_{rel}\exp\left[-NTU\left(1-C_{rel}\right)\right]}, & \text{if } C_{rel} < 1 \\ \dfrac{NTU}{1+NTU}, & \text{if } C_{rel} = 1 \end{cases}$
Parallel Flow	$\varepsilon = \dfrac{1-\exp\left[-NTU\left(1+C_{rel}\right)\right]}{1+C_{rel}}$

Shell and Tube	
One shell pass and two, four, or six tube passes	$$\varepsilon_1 = \frac{2}{1 + C_{rel} + \sqrt{1 + C_{rel}^2}\,\frac{1 + \exp\left(-NTU\sqrt{1 + C_{rel}^2}\right)}{1 - \exp\left(-NTU\sqrt{1 + C_{rel}^2}\right)}}$$
N Shell Passes and $2N$, $4N$, or $6N$ Tube Passes	$$\varepsilon = \frac{\left[\left(1 - \varepsilon_1 C_{rel}\right)/\left(1 - \varepsilon_1\right)\right]^N - 1}{\left[\left(1 - \varepsilon_1 C_{rel}\right)/\left(1 - \varepsilon_1\right)\right]^N - C_{rel}}$$
Cross Flow (Single Pass)	
Both Fluids Unmixed	$$\varepsilon = 1 - \exp\left(\frac{\exp\left(-C_{rel} NTU^{0.78}\right) - 1}{C_{rel} NTU^{-0.22}}\right)$$
Both Fluids Mixed	$$\varepsilon = \frac{1}{\frac{1}{1 - exp\left(-NTU\right)} + \frac{C_{rel}}{1 - \exp\left(-C_{rel} NTU\right)} - \frac{1}{NTU}}$$
C_{Max} mixed, C_{Min} unmixed	$$\varepsilon = \frac{1}{C_{rel}}\left(1 - \exp\left(-C_{rel}\left(1 - \exp\left(-NTU\right)\right)\right)\right)$$
C_{Max} unmixed, C_{Min} mixed	$$\varepsilon = 1 - \exp\left(-\frac{1}{C_{rel}}\left(1 - \exp\left(-C_{rel} NTU\right)\right)\right)$$

Assumptions and Limitations

- The heat exchanger fluids do not change phase. They are always thermal liquids.
- The heat exchanger is an adiabatic component. It does not transfer heat with its environment.

Parameters

Common Tab

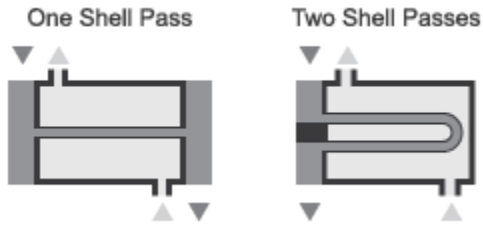
Flow arrangement

Heat exchanger geometry. Common geometries that you can select include `Parallel or counter flow`, `Shell and tube`, and `Cross flow`. Select `Generic — effectiveness table` to model other heat exchanger geometries based on tabular effectiveness data.

In the `Parallel or counter flow` configuration, the relative flow directions of fluids 1 and 2 determine whether the heat exchanger is based on parallel or counter flows. The flow directions depend on the remainder of the Simscape Fluids model.

Number of shell passes

Number of times the fluid in the tubes traverses the length of the shell before exiting. A single shell pass corresponds to a single traversal of the shell length between the tube inlet and outlet. The figure shows one- and two-pass examples.



One- and Two-Pass Configurations

This parameter is visible only when the **Flow arrangement** parameter is set to `Shell and tube`. The default value is `1`, corresponding to a single shell pass.

Cross flow type

Fluid mixing configuration. The fluids can be mixed or unmixed. The block uses the mixing configuration to determine which empirical heat transfer correlations to use. This parameter is visible only when the **Flow arrangement** parameter is set to `Cross flow`. The default setting is `Both fluids mixed`.

Number of heat transfer units vector, NTU

M-element vector of NTU values at which to specify the effectiveness tabular data. The number of transfer units (NTU) is a dimensionless parameter defined as

$$NTU = \frac{A_s U}{C_{min}},$$

where:

- A_S is the heat transfer surface area.
- U is the overall heat transfer coefficient.
- C_{min} is the smallest of the thermal capacity rates for the hot and cold fluids.

This parameter is visible only when the **Flow Arrangement** parameter is set to `Generic — effectiveness table`. The default vector is `[0.5, 1.0, 2.0, 3.0, 4.0]`.

Thermal capacity ratio vector, CR

N-element vector of thermal capacity ratios at which to specify the effectiveness tabular data. The thermal capacity ratio is the fraction

$$C_r = \frac{C_{min}}{C_{max}},$$

where C_{min} and C_{max} are the minimum and maximum thermal capacity rates. This parameter is visible only when the **Flow arrangement** parameter is set to `Generic — effectiveness table`. The default vector is `[0.0, 0.25, 0.5, 0.75, 1.0]`.

Effectiveness table, E(NTU, CR)

M-by-*N* matrix with the heat exchanger effectiveness values. The matrix rows correspond to the different values specified in the **Number of heat transfer units vector, NTU** parameter. The matrix columns correspond to the values specified in the **Thermal capacity ratio vector, CR** parameter.

This parameter is visible only when the **Flow arrangement** parameter is set to `Generic — effectiveness table`. The default table is a 6-by-5 matrix ranging in value from `0.30` to `0.99`.

Wall thermal resistance

Thermal resistance of the interface wall separating the two heat exchanger fluids. The block uses this parameter to compute the rate of heat transfer between the fluids. The default value is `1.6e-4` k/W.

Controlled Fluid 1 Tab | Controlled Fluid 2 Tab

Heat transfer surface area

Aggregate surface area for heat transfer between the cold and hot fluids. The default value is `0.01` m^2.

Fouling factor

Empirical parameter used to quantify the increased thermal resistance due to dirt deposits on the heat transfer surface. The default value is `1e-4` m^2*K/W.

Minimum fluid-wall heat transfer coefficient

Smallest allowed value of the heat transfer coefficient. The heat transfer coefficients specified through physical signal ports HC1 and HC2 saturate at this value. The default value is `5` W/(m^2*K).

The block uses the heat transfer coefficient to calculate the heat transfer rate between fluids 1 and 2 as described in “Heat Transfer Rate” on page 1-506.

Ports

- H1 — Thermal conserving port associated with the inlet temperature of fluid 1
- H2 — Thermal conserving port associated with the inlet temperature of fluid 2
- C1 — Physical signal input port for the thermal capacity rate of fluid 1
- C2 — Physical signal input port for the thermal capacity rate of fluid 2
- HC1 — Physical signal input port for the heat transfer coefficient between fluid 1 and the interface wall
- HC2 — Physical signal input port for the heat transfer coefficient between fluid 2 and the interface wall

References

[1] Holman, J. P. *Heat Transfer*. 9th ed. New York, NY: McGraw Hill, 2002.

[2] Shah, R. K. and D. P. Sekulic. *Fundamentals of Heat Exchanger Design*. Hoboken, NJ: John Wiley & Sons, 2003.

See Also

E-NTU Heat Exchanger (TL) | E-NTU Heat Exchanger (TL-TL)

Introduced in R2016a

Fan (G)

Mechanical apparatus for powering flow in gas systems
Library: Simscape / Fluids / Gas / Turbomachinery

Description

The Fan (G) block models a mechanical apparatus for powering flow in gas systems: a rotor mounted on a drive shaft (often) enclosed in a case. The fan is parameterized in terms of performance metrics commonly reported in technical data sheets (and specified here in tabulated form). The metrics used depend on the parameterization chosen but can include measures of pressure rise, flow rate, rotor speed, and fan efficiency. The parameterizations are valid solely in the normal operating region (for which data is reported).

Normally, an electrical motor, though sometimes another power source, spins the rotor (port **R**) against its case (**C**). The rotor transfers the power drawn to the gas stream, causing it to move (and its pressure to rise) from inlet (**A**) to outlet (**B**). The direction of the flow depends on the blades of the rotor and it can, in a real fan, be radial, axial, crossed, or mixed. Such effects are assumed to reflect entirely in the performance data specified for the fan.

The mechanical ports belong to the Simscape Rotational domain. To turn the rotor, they must connect to ports of the same domain. Rotational source blocks from the Simscape Foundation library are a simple way to apply the necessary torque. These are idealized models, without friction or other real-world effects. Simscape Driveline blocks are another option, richer in detail for more accurate simulations, among it the effects ignored in their idealized counterparts (when relevant to the model).

Mechanical Orientation

Two directions play into the block computations: those of the turning of the rotor and of the movement of the gas stream.

The shaft of the rotor can technically turn forward or backward. For the rotor to generate flow, however, only the direction indicated in the **Mechanical orientation** parameter will do. That direction can be `Positive` or `Negative`. (In other words, the fan is *unidirectional* in operation.) If the rotor shaft should turn in the counter direction, the fan goes into idle and power is no longer supplied to the stream. You can think of the rotor as having disengaged from the shaft (for example, by use of a one-way clutch).

The stream, on the other hand, when powered by the fan, must flow from inlet (port **A**) to outlet (**B**). This direction is designated as positive in the block calculations. It is possible for the flow to reverse, but not by the action of the rotor. Instead, an event must occur to flip the pressure gradient across the fan and so force the flow back toward the inlet. Such occurrences are considered abnormal and, when they take place, are generally transient and short-lived.

Note that the direction of the flow is independent of the **Mechanical orientation** setting of the block. This setting serves merely as a means to reverse the sweep of the rotor blades. Clockwise and counterclockwise rotors can both generate positive flow—and in fact must if the remainder of the fan requires it—but the way they must turn will differ. (One will generate positive flow when spun in the positive direction; the other, when spun in the negative direction.)

The saturation has the effect of splitting the rotor speed domain into two regions: below the saturation threshold, the rotor speed is fixed at the threshold value; above the saturation threshold, it is a variable determined by calculation. The transition between the regions has one drawback: without modification, it is sharp and its slope discontinuous.

Slope discontinuities pose a challenge to variable-step solvers (the sort commonly used in Simscape models). To precisely capture a sharp transition, the solver must reduce its time step, pausing briefly at the time of the transition in order to recompute the Jacobian matrix for the model (a representation of the dependencies between state variables and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present—it makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish a simulation. An alternative approach, used here, is to remove the slope discontinuities altogether, by smoothing them over a small range of motor speeds.

The smoothing, which adds a slight distortion to the transition, ensures that the fan eases into its saturated state rather than snap abruptly to it. The shape and scale of the smoothing derive from the cubic polynomial expression:

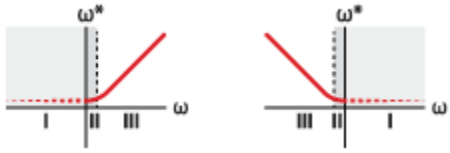
$$\lambda = 3\left(\frac{\omega}{\omega_{\mathrm{Th}}}\right)^2 - 2\left(\frac{\omega}{\omega_{\mathrm{Th}}}\right)^3,$$

where λ is the smoothing polynomial. The subscript Th denotes the threshold value at which to saturate the rotor speed (specified in the **Shaft speed threshold for flow reversal** block parameter). The smoothing adds a third, transitional, region between the saturated and variable regions of the rotor speed domain:

$$\omega^* = \begin{cases} w_{\mathrm{Th}}, & \omega < 0 \\ (1-\lambda)\,\omega_{\mathrm{Th}} + \lambda\omega, & \omega < \omega_{\mathrm{Th}}, \\ \omega, & \omega \geq \omega_{\mathrm{Th}} \end{cases}$$

where the asterisk (*) denotes the smoothed value.

The figure shows the effect of smoothing on the speed of the rotor. The plot to the left corresponds to a fan with positive mechanical orientation; the plot to the right, to one with negative mechanical orientation. Region **I** is the saturated state, region **II** the transition state, and region **III** the original state (with neither smoothing nor saturation).



Mass Balance

The fan, which creates flow but neither stores nor carries its contents, is a relatively compact device. The gas column that surrounds its rotor is thin and, on the scale of the larger components—the pipes, chambers, and reservoirs that often abut the fan—of negligible volume. This volume and its mass are ignored in this block. (While the stream can pass through the fan, its contents cannot accumulate there, as they might in a pipe, say, if it is dilated, heated, or compressed.)

Conservation of mass then requires only that the mass flow rates through the inlet and outlet be of the same magnitude:

$$\dot{m}_{\mathrm{A}} + \dot{m}_{\mathrm{B}} = 0,$$

where $\dot{m}$ is the mass flow rate *into* the valve through the port indicated by the subscript. Port **A** is always the inlet and port **B** always the outlet.

Energy Balance

The smaller the gas volume, the faster its transient response to pressure and temperature disturbances, and the shorter its time spent between steady states. If that volume is trivially small, as it is here, at least in proportion to the volumes of neighboring components, the transient response can be considered instantaneous. The fan then behaves as if it were always at a steady state (one given by the instantaneous conditions at the ports). Such components are known as *quasi-steady*.

A quasi-steady fan, having no gas volume, has also no energy contents to vary over time. Energy can flow through its ports, and, as mechanical work, from the rotor, but it cannot accumulate inside. In other words, what comes in must simultaneously go out. Conservation of energy then reduces to a sum of energy flow rates:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} + W_{\mathrm{F}} = 0,$$

where ϕ is the total energy flow rate *into* the fan through the port indicated by the subscript, and W_{F} is the rate of work done on—and the power supplied to—the stream by the rotor.

This is not to say that the fan is lossless. Energy can dissipate between the ports by interactions involving mechanical components, the gas stream, or both. The rotor shaft, for example, is subject to friction; the stream must pass through restrictions, elbows, and fittings. In this block, the losses due to all such factors are assumed to reflect in the total efficiency of the fan—the ratio of the (pneumatic) power output to the (mechanical) power input:

$$\eta_{\mathrm{T}} = \frac{W_{\mathrm{F}}}{W_{\mathrm{M}}},$$

where η_{T} is the total efficiency and W is the power delivered, in the term denoted by subscript `M`, by the drive shaft to the fan rotor (before any portion of that power can dissipate in the fan).

(The total efficiency is obtained during simulation from the tabulated data of the block. See "Fan Parameterizations" for the block calculations.)

The mechanical power input is defined in terms of the domain variables at the mechanical ports:

$$W_{\mathrm{M}} = \tau\omega^{*},$$

where τ is the torque delivered to the drive shaft (port **R**) relative to the case (**C**) and ω^{*} is the smoothed shaft velocity, also relative to the case, as defined in "Numerical Smoothing".

The power supplied to the stream, always a fraction η_{T} of that delivered by the drive shaft, acts to raise the total enthalpy of the gas from the inlet to the outlet. This power is defined here for ideal and nonideal gases alike as:

$$W_{\mathrm{F}} = \dot{m}\left(h_{\mathrm{T,B}} - h_{\mathrm{T,A}}\right),$$

where h_{T} is the specific total enthalpy at the outlet (subscript **B**) and at the inlet (subscript **A**). The specific total enthalpy is calculated from the specific enthalpy (the sum of specific internal energy with the product of pressure and specific volume):

$$h_{\mathrm{T}} = h + \frac{v^{2}}{2},$$

where the fraction on the right is the specific kinetic energy of the stream and v is the velocity of the same, both at the gas port being considered (**A** or **B**).

(The power W_{F} is frequently calculated in incompressible fluids as the product of the volumetric flow rate through and the pressure rise across the fan; the use of a compressibility factor allows a variant of the same expression to be used in ideal gases but, for nonideal gases, especially those subjected to large pressure rises, neither expression will give accurate results).

Inlet Density Definition

The fan parameterizations rely partly on the so-called fan affinity laws. These are expressions of proportionality used to relate fans of the same type but of different size and at different operating conditions—reflected partly in the density of the gas.

As the fan is modeled without gas volume, that density is obtained from the nearest upstream computational node. That node is port **A** during normal operation and port **B** if the stream should be forced in the reverse direction:

$$\rho = \begin{cases} \rho_{\mathrm{A}}, & \dot{m} > 0 \\ \rho_{\mathrm{B}}, & \dot{m} > 0 \end{cases},$$

where ρ is the gas density at the port indicated by the subscript.

For numerical robustness, when the mass flow rate drops below a small threshold, the gas density is artificially smoothed between the values at the ports. The smoothing is based on the transcendental function:

$$\alpha = \tanh\left(\frac{4\dot{m}}{\dot{m}_{\mathrm{Th}}}\right),$$

where α is the smoothing factor and `tanh` is the hyperbolic tangent function. The subscript `Th` denotes a threshold value (a very small number hardcoded in the block). In terms of the smoothing factor, in the transitional region below the mass flow rate threshold, the gas density becomes:

$$\rho = \rho_{\mathrm{A}}\left(\frac{1+\alpha}{2}\right) + \rho_{\mathrm{B}}\left(\frac{1-\alpha}{2}\right).$$

This expression adds a third row to the piecewise definition of gas density, which is finally defined as:

$$\rho^{*} = \begin{cases} \rho_{\mathrm{A}}, & \dot{m} > \dot{m}_{\mathrm{Th}} \\ \rho_{\mathrm{A}}\left(\frac{1+\alpha}{2}\right) + \rho_{\mathrm{B}}\left(\frac{1-\alpha}{2}\right), & |\dot{m}| \le \dot{m}_{\mathrm{Th}} \\ \rho_{\mathrm{B}} & \dot{m} < -\dot{m}_{\mathrm{Th}} \end{cases}.$$

The asterisk denotes a smoothed value. The mass flow rate is the same at port **A** and port **B**. The value used in the conditions of the gas density expression can refer to either port.

Fan Parameterizations

The relationships between pressure rise, flow rate, fan efficiency, and rotor speed derive from the tabulated data specified in the block. The form of this data and the calculations that it supports depend on the fan parameterization chosen. Of parameterization options there are four. One is simple and based on one-dimensional tables; three are more complete and based on two-dimensional tables. All are restricted to the normal operating region (given in the technical data sheets of the fan).

The pressure rise is calculated from the second fan affinity law, which, modified for fans of the same size, states:

$$p_{\mathrm{SF}} = \left(\frac{\omega^*}{\omega_{\mathrm{R}}}\right)^2 \left(\frac{\rho^*}{\rho_{\mathrm{R}}}\right) p_{\mathrm{SF,R}}.$$

The subscript R denotes a reference value (and the asterisk a smoothed quantity, as described previously). The reference density and the reference shaft speed are each obtained from a block parameter (of the same name). The static pressure rise at reference conditions is obtained from the tabulated data of the block:

$$p_{\mathrm{SF,R}} = p_{\mathrm{SF}}(q_{\mathrm{R}}),$$

where q_{R} is the volumetric flow rate at reference conditions (and the independent variable in the tabulated data). Its value is calculated from the first fan affinity law:

$$q_{\mathrm{R}} = \left(\frac{\omega_{\mathrm{R}}}{\omega^*}\right) q,$$

where q is the volumetric flow rate through the fan, calculated from the mass flow rate at the ports as:

$$q = \frac{\dot{m}}{\rho^*}.$$

(The first fan law is used here to capture the effect of shaft speed on volumetric flow rate (and therefore also static pressure rise). This calculation is omitted in the `2D tabulated data - static pressure and total efficiency vs. shaft speed and flow`

`rate` parameterization, where the effect of shaft speed is taken into account directly in the tabulated data of the block.)

The total fan efficiency, described in "Total Efficiency", is likewise obtained from the tabulated data:

$$\eta_T = \eta_T(q_R).$$

With the rotor shaft speed now specified in the tabulated data, its reference value can be set to its instantaneous (and smoothed) value. The ratio of actual to reference shaft speeds reduces to `1` and the second affinity law gives:

$$p_{S,F} = \left(\frac{\rho^*}{\rho_R}\right) p_{SF,R},$$

where $p_{SF,R}$ is obtained from the tabulated data as a bivariate function of the (smoothed) rotor shaft speed and of the volumetric flow rate through the fan:

$$p_{SF,R} = p_{SF}(\omega, q).$$

The total fan efficiency is obtained in a similar manner from the tabulated data:

$$\eta_T = \eta_T(\omega, q).$$

The mass flow rate through the fan is obtained from the tabulated data of the block as:

$$\dot{m} = \rho_R q_R,$$

where q_R is the volumetric flow rate through the fan at reference density:

$$q_R = q(\omega, p_{SF,R}),$$

The static pressure rise at the same reference condition is calculated as:

$$p_{SF,R} = \left(\frac{\rho_R}{\rho^*}\right) p_{SF}.$$

The total fan efficiency is defined as before:

$$\eta_T = \eta_T(\omega, p_{SF,R}).$$

The mass flow rate is obtained from the tabulated data of the block as:

$$\dot{m} = \rho_R q_R,$$

where the volumetric flow rate at reference density is computed no longer from the static pressure rise but from its ratio to its maximum rated value:

$$q_R = q(\omega, r_{p,R}).$$

The static pressure rise ratio, r_p, is specified in the tabulated data of the block. It is defined at reference density as:

$$r_{p,R} = \frac{p_{SF,R}}{p_{SF,Max}},$$

which, using the first fan affinity law, becomes:

$$r_{p,R} = \left(\frac{\rho_R}{\rho^*}\right)\left(\frac{p_{SF}}{p_{SF,Max}}\right),$$

The maximum static pressure rise, $p_{SF,Max}$, can vary with the rotor shaft speed. The relationship between the two is specified in a separate, one-dimensional lookup table. A plot of flow rate against static pressure rise need not be rectangular in shape, giving this parameterization a versaility not found in its alternatives.

The total fan efficiency is defined in terms of the same variables as:

$$\eta_T = \eta_T(\omega, r_p).$$

Ports

Conserving

A — Inlet
gas

Opening for the gas stream to enter the fan.

B — Outlet
gas

Opening for the gas stream to exit the fan.

R — Rotor
gas

Drive shaft on which the fan rotor is to be mounted.

C — Case
gas

Case in which the fan rotor is to be housed.

Parameters

Fan specification — Set of performance metrics on which to base the fan model
`1D tabulated data - static pressure and total efficiency vs. flow rate` (default) | `2D tabulated data - static pressure and total efficiency vs. shaft speed and flow rate` | `2D tabulated data - flow rate and total efficiency vs. shaft speed and static pressure` | `2D tabulated data - flow rate and total efficiency vs. shaft speed and static pressure ratio`

Set of performance metrics on which to base the fan model. These metrics are commonly reported in technical data sheets as plots or tables. The 1-D parameterization uses the first fan affinity law to obtain the rotor shaft speed data. (See the block description for the affinity laws.)

`Mechanical orientation` — Turning direction required of the fan rotor to generate flow
`Positive` (default) | `Negative`

Direction in which the fan rotor must turn in order to generate flow. The fan is unidirectional and the gas stream flows only in the positive direction in each case. Use this parameter to capture different rotor designs—for example, those with forward-swept and backward-swept blades, which must turn in different directions relative to the flow.

`Volumetric flow rate vector` — Volumetric flow rates at which to tabulate the active fan characteristics
`[4.7, 5.2, 5.7, 6.1, 6.6] m^3/s` (default) | vector in units of volume/time

Volumetric flow rates, as measured from inlet to outlet, at which to specify the active fan characteristics (those used by the selected parameterization). These include static pressure and total efficiency for the fan, specified either as 1-D or 2-D tables. The default vector depends on the parameterization of the block.

The number of elements in the vector must equal the number of elements in the 1-D table or the number of rows in the 2-D table. (The tables are obtained from the **Static pressure rise vector** and **Static pressure rise vector** parameters or from the **Static pressure rise table** and **Total efficiency table** parameters. The vector elements must increase monotonically from left to right.

The fan generates flow in the forward, or positive, direction only. Negative values, which are counter to this direction, are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `1D tabulated data - static pressure and total efficiency vs. flow rate`.

`Static pressure rise vector` — Static pressure differentials from inlet to outlet at tabulated volumetric flow rates
`[1120, 995.36, 870.94, 746.52, 622.1]` (default) | unitless vector

Static pressure differentials from inlet to outlet at the specified volumetric flow rates. The number of elements in this vector must equal that in the **Volumetric flow rate vector** parameter. The fan generates flow in the forward direction only (from inlet to outlet). Negative static pressure differentials, which generally map to reverse flows, are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `1D tabulated data - static pressure and total efficiency vs. flow rate`.

`Total efficiency vector` — Total fan efficiencies at tabulated volumetric flow rates

`[.1, .2, .25, .15, .1]` (default) | unitless vector

Total fan efficiencies at the specified volumetric flow rates. The total efficiency is the ratio of the output power (transferred to the gas stream) to the inlet power (supplied by the rotor shaft).

As is typical of efficiency parameters, this parameter is defined as a fraction between `0` and `1`. The number of elements in this vector must equal that in the **Volumetric flow rate vector** parameter. Negative values, which imply a fan capable of extracting power from (instead of supplying it to) the gas stream are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `1D tabulated data - static pressure and total efficiency vs. flow rate`.

`Shaft speed vector, omega` — Rotor shaft speeds at which to tabulate the active fan characteristics

`[450, 500, 550, 600, 650] rpm` (default) | vector in units of rotation/time

Rotor shaft speeds, measured against the fan case, at which to tabulate the active fan characteristics. Those vary with the fan parameterization but include total efficiency and either static pressure of flow rate. The default vector depends on the parameterization selected.

The number of elements in the vector must equal the numbers of rows in the tables of fan characteristics. (The tables are obtained from the **Static pressure rise table**, **Total efficiency table**, and **Flow rate table** block parameters, whichever happen to be active.)

The vector elements must increase monotonically in value from left to right. Negative values, which imply a fan capable of powering flow in reverse, are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to a 2-D parameterization.

`Static pressure rise vector, Dp` — Static pressure differentials from inlet to outlet at which to tabulate the active fan characteristics

`[1 / 8, 1 / 4, 3 / 8, 1 / 2] * 3386.4 Pa` (default) | vector in units of pressure

Static pressure differentials, as measured from inlet to outlet, at which to specify the active fan characteristics (those used by the selected parameterization). Here, those characteristics include flow rate and total efficiency.

Static pressure is that measured when the dynamic pressure, generally due to flow, is zero or subtracted from the (total) pressure reading. Fan characteristics are commonly reported in terms of it, though some manufacturers use the total pressure instead.

The number of elements in the vector must equal the numbers of rows in the tables of fan characteristics. (The tables are obtained from the **Flow rate table** and **Total efficiency table** parameters.) The vector elements must increase monotonically in value from left to right. Negative values, which imply a fan capable of powering flow in reverse, are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `2D tabulated data - flow rate and total efficiency vs. shaft speed and static pressure`.

`Static pressure rise ratio vector, Dp/DpMax` — Normalized static pressure differentials from inlet to outlet at which to tabulate the active fan characteritiscs

`linspace(0, 1, 7)` (default) | unitless vector

Normalized static pressure differentials, as measured from inlet to outlet, at which to specify the active fan characteristics (those used by the selected parameterization). Here, those characteristics include flow rate and total efficiency.

The normalization is with respect to the maximum pressure rise allowed at a given volumetric flow rate (and computed during simulation from the same). In other words, this parameter is the fraction of the static pressure rise over its calculated maximum. See the block description for more detail on this parameter.

The number of elements in the vector must equal the numbers of rows in the tables of fan characteristics. (The tables are obtained from the **Flow rate table** and **Total efficiency table** parameters.) The vector elements must increase monotonically in value from left to right. Negative values, which imply a fan capable of powering flow in reverse, are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `2D tabulated data - flow rate and total efficiency vs. shaft speed and static pressure ratio`.

`Static pressure rise table, Dp(omega,q)` — Static pressure differentials from inlet to outlet at specified breakpoints

`[675, 650, 560, 420; 830, 800, 695, 520; 1008, 966, 840, 630; 1200, 1150, 1000, 750; 1370, 1300, 1250, 1000]` `Pa` (default) | vector in units of pressure

Static pressure differentials, as measured from inlet to outlet, at the breakpoints specified in the block. (The breakpoints are given in this parameterization by the **Shaft speed vector** and **Flow rate vector** block parameters.)

Static pressure is that measured when the dynamic pressure, generally due to flow, is zero or subtracted from the (total) pressure reading. Fan characteristics are commonly reported in terms of it, though some manufacturers use the total pressure instead.

The number of rows in the table must equal the number of elements in the first vector specified (**Shaft speed vector**); the number of columns, the number of elements in the second vector specified (**Flow rate vector**). The fan generates flow in the forward, or positive, direction only. Negative pressure differentials, which could force the flow in reverse, are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `2D tabulated data - static pressure and total efficiency vs. shaft speed and flow rate`.

`Total efficiency table, Eta(omega,q)` — Total fan efficiencies at specified breakpoints

`[.22, .23, .25, .24; .38, .39, .4, .34; .58, .6, .62, .6; .81, .82, .84, .8; .88, .91, .95, .9]` (default) | unitless vector

Total fan efficiencies at the breakpoints specified in the block. (The variables used to define the breakpoints depend on the fan parameterization; they can include shaft speed, flow rate, static pressure, and/or static pressure ratio.)

The total efficiency is the ratio of the output power (transferred to the gas stream) to the inlet power (supplied by the rotor shaft). As is typical of efficiency parameters, this parameter is defined as a fraction, normally between `0` and `1`.

The number of rows in the table must equal the number of elements in the first vector specified (**Shaft speed vector**); the number of columns, the number of elements in the second vector specified (**Flow rate vector**, **Static pressure rise vector**, or **Static pressure rise ratio vector**).

Negative values, which imply a fan capable of extracting power from (instead of supplying it to) the gas stream are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to a 2-D parameterization.

`Flow rate table, q(omega,Dp)` — Volumetric flow rates at specified breakpoints

`[.3282, .2965, .2556, .1973; .8254, .71, .5876, .4545; 1.2018, 1.0829, .9083, .7034; 1.5065, 1.4098, 1.2722, .9844; 1.8368, 1.7507, 1.6055, 1.2417] m^3/s` (default) | vector in units of volume/time

Volumetric flow rates, measured from inlet to outlet, at the breakpoints specified in the block. (The variables used to define the breakpoints depend on the fan parameterization; they can include shaft speed, static pressure rise, and static pressure rise ratio.)

The number of rows in the table must equal the number of elements in the first vector specified (**Shaft speed vector**); the number of columns, the number of elements in the second vector specified (**Static pressure rise vector**, or **Static pressure rise ratio vector**).

The fan generates flow in the forward, or positive, direction only. Negative values, which are counter to this direction, are disallowed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `2D tabulated data - flow rate and total efficiency`

`vs. shaft speed and static pressure` or `2D tabulated data - flow rate and total efficiency vs. shaft speed and static pressure ratio`.

`Reference density` — Gas density established in the measurement of tabulated reference data

`1.2 kg/m^3` (default) | positive scalar in units of mass/volume

Gas density established in the measurement of the tabulated reference data specified—the volumetric flow rate, static pressure rise, and total efficiency for the fan. This parameter is used in the calculation of the true static pressure rise across the fan.

`Reference shaft speed` — Rotor shaft speed used in the measurement of tabulated reference data

`910 rpm` (default) | positive scalar in units of rotation/time

Rotor shaft speed used in the measurement of the tabulated reference data specified—the volumetric flow rate, static pressure rise, and total efficiency for the fan. This parameter is used in the calculation of the true static pressure rise across the fan.

Dependencies

This parameter is active and exposed in the block dialog box when the **Fan specification** parameter is set to `1D tabulated data - static pressure and total efficiency vs. flow rate`.

`Shaft speed threshold for flow reversal` — Minimum rotor shaft speed required to generate flow

`1 rpm` (default) | positive scalar in units of rotation/time

Lower bound of the rotor shaft speed range at which the fan will generate flow. This parameter is always positive, no matter the **Mechanical orientation** setting.

For use in the flow rate calculations, the rotor speed obtained from ports **R** and **C** is saturated at this value. The saturation is smooth, with the degree of smoothing dependent partly on this parameter. The smoothing is based on a cubic polynomial (given in the block description).

`Inlet gas flow area (port A)` — Area normal to the flow at the inlet of the fan

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow at the inlet of the fan. The inlet and outlet need not be the same in size. For best results, the value specified here should match the opening area of the component connected to the inlet.

`Outlet gas flow area (port B)` — Area normal to the flow at the outlet of the fan

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow at the outlet of the fan. The inlet and outlet need not be the same in size. For best results, the value specified here should match the opening area of the component connected to the outlet.

See Also

Variable Orifice ISO 6358 (G)

Introduced in R2018b

Fixed Orifice

Hydraulic orifice with constant cross-sectional area

Library

Orifices

Description

The Fixed Orifice block models a sharp-edged constant-area orifice, flow rate through which is proportional to the pressure differential across the orifice. The flow rate is determined according to the following equations:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

$$\Delta p = p_A - p_B,$$

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
A	Orifice passage area
ρ	Fluid density
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

$$p_{avg} = (p_A + p_B)/2$$

where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Orifice hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as

$\Delta p = p_A - p_B$, .

Variables

Use the **Variables** tab to set the priority and initial target values for the block variables prior to simulation. For more information, see “Set Priority and Initial Target for Block Variables” (Simscape).

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.

Parameters

Orifice area

Orifice passage area. The default value is `1e-4` m^2.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter

value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the orifice inlet.

`B`

Hydraulic conserving port associated with the orifice outlet.

See Also

Annular Orifice | Constant Area Hydraulic Orifice | Fixed Orifice with Fluid Inertia | Fixed Orifice Empirical | Orifice with Variable Area Round Holes | Orifice with Variable Area Slot | Variable Area Hydraulic Orifice | Variable Orifice

Introduced in R2006a

Fixed Orifice Empirical

Fixed hydraulic orifice based on empirical data

Library

Orifices

Description

The Fixed Orifice Empirical block models a fixed orifice in hydraulic systems with data obtained from experimental testing. The data required for characterization is the set of pressure differentials across the orifice, together with associated flow rates determined for a particular fluid and temperature.

The flow rate through the orifice is determined with the table lookup block from the set of experimental measurements

$$q = f(p)$$

where

q	Volumetric flow rate
p	Pressure differential

You can select from two interpolation and two extrapolation methods.

Note Extrapolation could result in large errors if the operating point deviates significantly from the area of experiments.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as

$\Delta p = p_A - p_B$, .

Parameters

Tabulated pressure differentials

Specify the pressure differential vector as a one-dimensional array. The vector contains values of pressure differentials obtained during experimental characterization of the orifice. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in bar, are `[-40.0, -30.0, -20.0, -10.0, -5.0, 0.0, 5.0, 10.0, 20.0, 30.0, 40.0]`.

Tabulated flow rates

Specify the vector of flow rates through the orifice, corresponding to these pressure differentials, as a one-dimensional array. The vector must be of the same size as the **Tabulated pressure differentials** vector. The default values, in m^3/s, are `[ -0.0024436 -0.0021248 -0.0016852 -0.0012248 -8.4476e-4 0 8.5333e-4 0.0012068 0.0017067 0.0020902 0.0024136 ]`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page.

Extrapolation method

Use extrapolation with extreme caution, because moving beyond the measurement limits could result in large errors. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the orifice inlet.

`B`

Hydraulic conserving port associated with the orifice outlet.

See Also

Fixed Orifice | Fixed Orifice with Fluid Inertia

Introduced in R2012b

Fixed Orifice with Fluid Inertia

Fixed hydraulic orifice accounting for flow inertia

Library

Orifices

Description

The Fixed Orifice with Fluid Inertia block models a hydraulic fixed orifice and accounts for the fluid inertia, in addition to the static pressure loss.

Fluid inertia plays a noticeable role in orifices with a large ratio of orifice length to the orifice hydraulic diameter (L / D_H) and in sharp-edged short orifices when the rate of change of flow rate (fluid acceleration) is relatively large.

The orifice is based on the following equations:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p_r}{\left(p_r^2 + p_{cr}^2\right)^{1/4}}$$

$$p = p_{in} + p_r$$

$$p_{in} = \rho \frac{L}{A} \frac{dq}{dt}$$

where

q	Volumetric flow rate

p	Total pressure differential
p_{in}	Inertial pressure drop
p_r	Resistive pressure drop
C_D	Flow discharge coefficient
A	Orifice passage area
L	Orifice length
ρ	Fluid density
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

  $$p_{avg} = (p_A + p_B)/2$$

  where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

  $$D_H = \sqrt{\frac{4A}{\pi}}$$

  where

D_H	Orifice hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

Connections A and B are conserving hydraulic ports associated with the orifice inlet and outlet, respectively. The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $\Delta p = p_A - p_B$, .

Parameters

Orifice area

Cross-sectional area of the orifice. The default value is `1e-4` m^2.

Orifice length

Total length of the orifice. Generally, increase the geometrical length of the orifice up to $2 \cdot 0.8 \cdot D_H$ (where D_H is the orifice hydraulic diameter) to take into account the added volumes of fluid on both sides of the orifice. The default value is `0.01` m.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. The coefficient affects the resistive pressure drop in the orifice. The default value is `0.6`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `10`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Initial flow rate

Flow rate through the orifice at the start of simulation. This parameter specifies the initial condition for use in computing the block's state at the beginning of a simulation run. For more information, see “Initial Conditions Computation” (Simscape). The default value is `0`.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the orifice inlet.

`B`

Hydraulic conserving port associated with the orifice outlet.

See Also

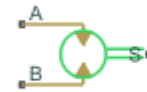
Fixed Orifice | Fixed Orifice Empirical

Introduced in R2013a

Fixed-Displacement Motor

Hydraulic-to-mechanical power conversion device

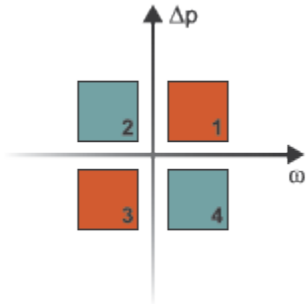
Library: Simscape / Fluids / Hydraulics (Isothermal) / Pumps and Motors



Description

The Fixed-Displacement Motor block represents a device that extracts power from a hydraulic (isothermal liquid) network and delivers it to a mechanical rotational network. The motor displacement is fixed at a constant value that you specify through the **Displacement** parameter.

Ports **A** and **B** represent the motor inlet and outlet, respectively. Port **S** represents the motor drive shaft. During normal operation, the angular velocity at port **S** is positive if the pressure drop from port **A** to port **B** is positive also. This operation mode is referred to here as forward motor.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure drop from port **A** to port **B** (Δp) and on the angular velocity at port **S** (ω). The “Operation

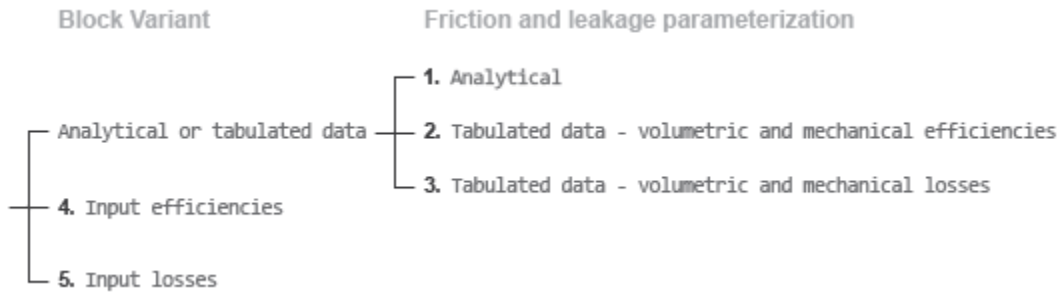
Modes” on page 1-542 figure maps the modes to the octants of a Δ*p*-*ω*-*D* chart. The modes are labeled 1–4:

- Mode **1**: forward motor — A positive pressure drop generates a positive shaft angular velocity.
- Mode **2**: reverse pump — A negative shaft angular velocity generates a negative pressure gain (shown in the figure as a positive pressure drop).
- Mode **3**: reverse motor — A negative pressure drop generates a negative shaft angular velocity.
- Mode **4**: forward pump — A positive shaft angular velocity generates a positive pressure gain (shown in the figure as a negative pressure drop).

The response time of the motor is considered negligible in comparison with the system response time. The motor is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The motor model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the motor inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink® variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Flow Rate and Driving Torque

The volumetric flow rate required to power the motor is

$$q = q_{\text{Ideal}} + q_{\text{Leak}},$$

where:

- q is the net volumetric flow rate.
- q_{Ideal} is the ideal volumetric flow rate.
- q_{Leak} is the internal leakage volumetric flow rate.

The torque generated at the motor is

$$\tau = \tau_{\text{Ideal}} - \tau_{\text{Friction}},$$

where:

- τ is the net torque.
- τ_{Ideal} is the ideal torque.
- τ_{Friction} is the friction torque.

The ideal volumetric flow rate is

$$q_{\text{Ideal}} = D\omega,$$

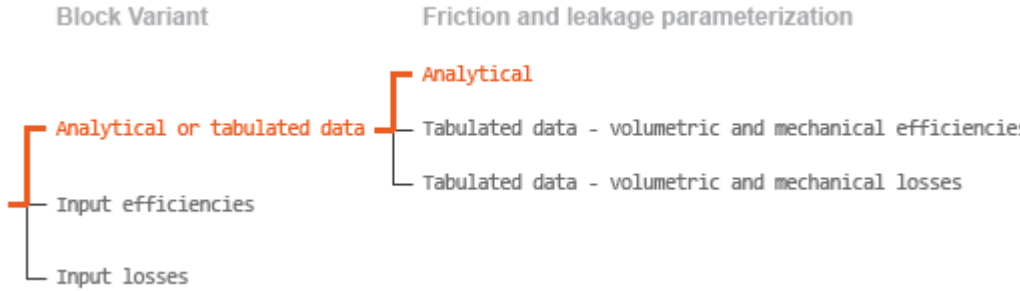
and the ideal generated torque is

$$\tau_{\text{Ideal}} = D\Delta p,$$

where:

- D is the specified value of the **Displacement** block parameter.
- ω is the instantaneous angular velocity of the rotary shaft.
- Δp is the instantaneous pressure drop from inlet to outlet.

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$q_{\text{Leak}} = K_{\text{HP}}\Delta p,$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}}|\Delta p|\right)\tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

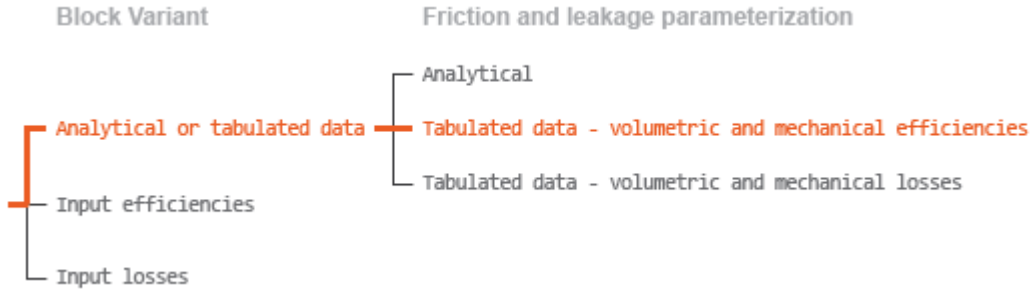
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- $\omega_{Threshold}$ is the threshold angular velocity for the motor-pump transition. The threshold angular velocity is an internally set fraction of the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{HP} = \frac{\nu_{Nom}}{\rho v}\frac{\rho_{Nom}\omega_{Nom}D_{Max}}{\Delta p_{Nom}}\left(\frac{1}{\eta_{v,Nom}} - 1\right),$$

where:

- ν_{Nom} is the specified value of the **Nominal kinematic viscosity** block parameter. This is the kinematic viscosity at which the nominal volumetric efficiency is specified.
- ρ_{Nom} is the specified value of the **Nominal fluid density** block parameter. This is the density at which the nominal volumetric efficiency is specified.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- ρ is the actual fluid density in the attached hydraulic (isothermal liquid) network. This density can differ from the specified value of the **Nominal fluid density** block parameter.
- ν is the kinematic viscosity of the fluid associated with the fluid network.
- Δp_{Nom} is the specified value of the **Nominal pressure drop** block parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{v,Nom}$ is the specified value of the **Volumetric efficiency at nominal conditions** block parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`, the leakage flow rate is

$$q_{\text{Leak}} = q_{\text{Leak,Motor}} \frac{(1+\alpha)}{2} + q_{\text{Leak,Pump}} \frac{(1-\alpha)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Motor}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Pump}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the motor-pump transition.
- $q_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $q_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure drop threshold for motor-pump transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.

The leakage flow rate is computed from efficiency tabulated data through the equation

$$q_{\text{Leak,Motor}} = (1-\eta_{\text{v}})\,q,$$

in motor mode and through the equation

$$q_{\text{Leak,Pump}} = -(1-\eta_{\text{v}})\,q_{\text{Ideal}}.$$

in pump mode, where:

- η_{v} is the volumetric efficiency obtained through interpolation or extrapolation of the **Volumetric efficiency table, e_v(dp,w)** parameter data.

Similarly, the friction torque is computed from efficiency tabulated data through the equation

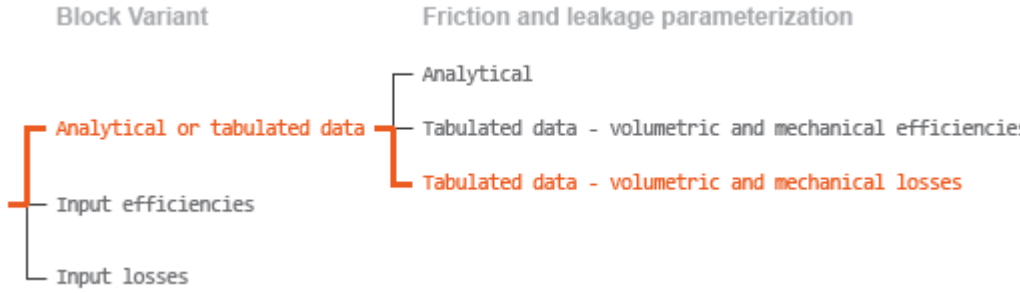
$$\tau_{\text{Friction,Motor}} = (1-\eta_{\text{m}})\,\tau_{\text{Ideal}},$$

in motor mode and through the equation

$$\tau_{\text{Friction,Pump}} = -(1-\eta_{\text{m}})\,\tau.$$

in pump mode, where:

- η_{m} is the mechanical efficiency obtained through interpolation or extrapolation of the **Mechanical efficiency table, e_m(dp,w)** parameter data.



Case 3: Loss Tabulated Data

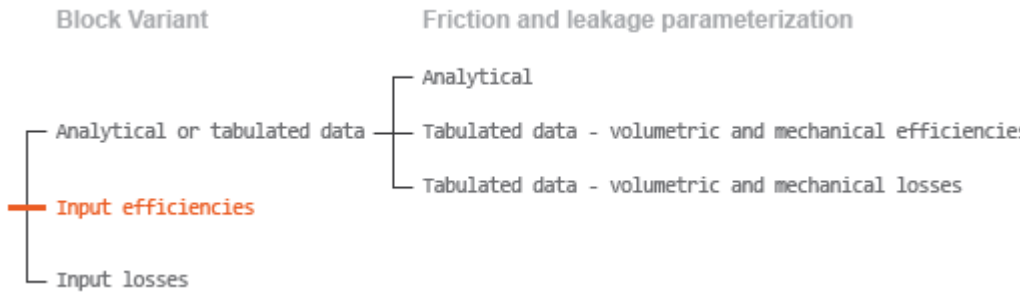
If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical losses`, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}\left(\Delta p, \omega\right).$$

and the friction torque equation is

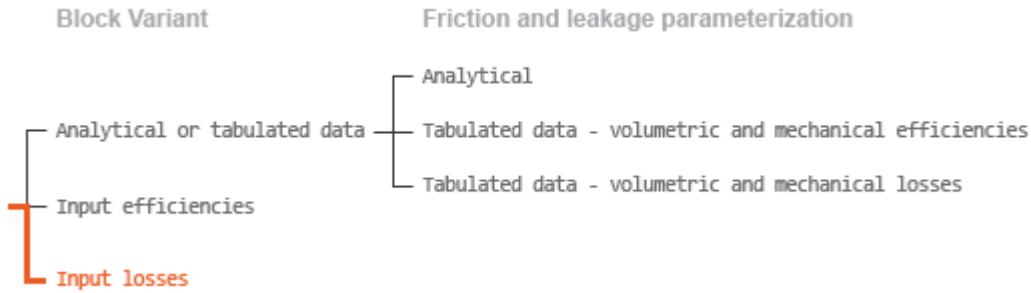
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega\right),$$

where $q_{\text{Leak}}(\Delta p,\omega)$ and $\tau_{\text{Friction}}(\Delta p,\omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the **Volumetric loss table, q_loss(dp,w)** and **Mechanical loss table, torque_loss (dp,w)** parameter data.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

Characteristic Curve Visualization

If the block variant is set to `Analytical or tabulated data`, you can plot a variety of performance, efficiency, and loss curves from simulation data and component parameters. Use the context-sensitive menu of the block to plot the characteristic curves. Right-click the block to open the menu and select **Fluids > Plot characteristic**. A test harness opens with instructions on how to generate the curves. See “Pump and Motor Characteristic Curves”.

Assumptions

- Fluid compressibility is negligible.
- Loading on the motor shaft due to inertia, friction, and spring forces is negligible.

Ports

Input

EV — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

EM — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

LV — Volumetric loss, `m^3/s`

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the motor inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

LM — Mechanical loss, `N*m`

physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotating motor shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

A — Motor inlet
hydraulic (isothermal)

Hydraulic (isothermal liquid) port representing the motor inlet.

B — Motor outlet
hydraulic (isothermal)

Hydraulic (isothermal liquid) port representing the motor outlet.

S — Motor shaft
mechanical rotational

Mechanical rotational port representing the motor shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-543.

Variant 1: Analytical or tabulated data

`Displacement` — Fluid volume displaced per unit shaft rotation angle
`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque
`Analytical` (default)

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup

tables to map pressure drop and angular velocity to component efficiencies or losses. The tabular options include:

- `Tabulated data – volumetric and mechanical efficiencies`
- `Tabulated data – volumetric and mechanical losses`

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data`.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the nominal volumetric efficiency

`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotating shaft at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure drop` — Pressure gain at which to specify the nominal volumetric efficiency

`100e5 Pa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal kinematic viscosity` — Kinematic viscosity at which to specify the nominal volumetric efficiency
`18 cSt` (default) | scalar with units of area/time

Kinematic viscosity of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal fluid density` — Fluid density at which to specify the nominal volumetric efficiency
`900 kg/m^3` (default) | scalar with units of mass/volume

Mass density of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions
`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation
`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure drop coefficient` — Proportionality constant between friction torque and pressure drop
`0.6e-6 N*m/Pa` (default) | scalar with units of torque/pressure

Proportionality constant between the friction torque on the mechanical shaft and the pressure drop from inlet to outlet.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

`Pressure drop vector for efficiencies, dp` — Pressure drops at which to specify the volumetric and mechanical efficiencies
M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies
N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Volumetric efficiency table, e_v(dp,w)` — Volumetric efficiencies at the specified pressure drops and angular velocities
unitless *M*-by-*N* matrix

M-by-*N* matrix with the volumetric efficiencies at the specified fluid pressure drops and shaft angular velocities. The efficiencies must fall in the range of 0–1. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w)` — Mechanical efficiencies at the specified pressure drops and angular velocities

unitless *M*-by-*N* matrix

M-by-*N* matrix with the mechanical efficiencies corresponding to the specified fluid pressure drops and shaft angular velocities. The efficiencies must fall in the range of 0–1. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motoring and pumping modes

`1e5 Pa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Angular velocity threshold for pump–motor transition` — Shaft angular velocity at which to initiate a smooth transition between motoring and pumping modes

`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Check if operating beyond the quadrants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data

`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure gain or shaft angular velocity cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies` or `Tabulated data – volumetric and mechanical losses`.

`Pressure drop vector for losses, dp` — Pressure drops at which to specify the volumetric and mechanical losses

M-element vector with units of pressure (default)

M-element vector of pressure drops at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses

N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the component loss data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Volumetric loss table, q_loss(dp,w)` — Internal leakage flow rates at the specified pressure gains and angular velocities

M-by-*N* matrix with units of volume/time

M-by-*N* matrix with the volumetric gains at the specified fluid pressure drops and shaft angular velocities. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Mechanical loss table, torque_loss(dp,w)` — Friction torques at the specified pressure gains and angular velocities

M-by-*N* matrix with units of torque

M-by-*N* matrix with the mechanical losses at the specified fluid pressure drops and shaft angular velocities. Mechanical loss is defined here as the friction torque due to seals and internal components. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Displacement` — Fluid volume displaced per unit shaft rotation angle

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motoring and pumping modes

`1e5 Pa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for motor–pump transition` — Shaft angular velocity at which to initiate a smooth transition between motoring and pumping modes

`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at this value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure

`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning

`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

Variant 3: Input losses

`Displacement` — Fluid volume displaced per unit shaft rotation angle

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Check if operating beyond the motor mode` — Simulation warning mode for operating conditions outside motoring mode

`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the motoring mode. A warning is issued if the motor transitions to pumping mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Input losses`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure

`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning

`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

See Also

Fixed-Displacement Motor (TL) | Fixed-Displacement Pump | Fixed-Displacement Pump (TL) | Variable-Displacement Motor | Variable-Displacement Pump

Introduced in R2006b

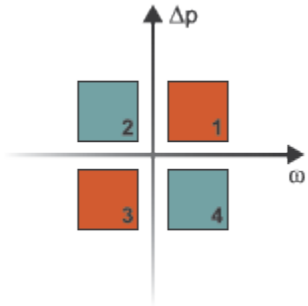
Fixed-Displacement Motor (TL)

Hydraulic-mechanical power conversion device
Library: Simscape / Fluids / Thermal Liquid / Pumps & Motors

Description

The Fixed-Displacement Motor (TL) block represents a device that extracts power from a hydraulic (isothermal liquid) network and delivers it to a mechanical rotational network. The motor displacement is fixed at a constant value that you specify through the **Displacement** parameter.

Ports **A** and **B** represent the motor inlets. Ports **R** and **C** represent the motor drive shaft and case. During normal operation, a pressure drop from port **A** to port **B** causes a positive flow rate from port **A** to port **B** and a positive rotation of the motor shaft relative to the motor case. This operation mode is referred to here as *forward motor*.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure drop from port **A** to port **B** (Δp) and on the angular velocity at port **R** relative to port **C**

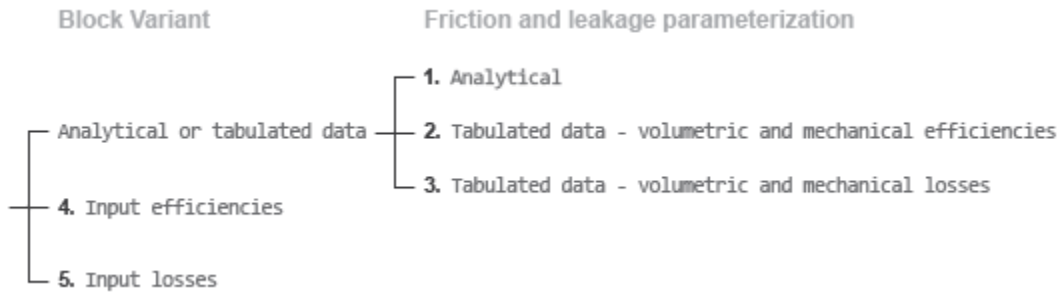
(ω). The “Operation Modes” on page 1-542 figure maps the modes to the octants of a Δp-ω-D chart. The modes are labeled 1–4:

- Mode **1**: forward motor — A positive pressure drop generates a positive shaft angular velocity.
- Mode **2**: reverse pump — A negative shaft angular velocity generates a negative pressure gain (shown in the figure as a positive pressure drop).
- Mode **3**: reverse motor — A negative pressure drop generates a negative shaft angular velocity.
- Mode **4**: forward pump — A positive shaft angular velocity generates a positive pressure gain (shown in the figure as a negative pressure drop).

The response time of the motor is considered negligible in comparison with the system response time. The motor is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The motor model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the motor inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Flow Rate and Driving Torque

The mass flow rate generated at the motor is

$$\dot{m} = \dot{m}_{\mathrm{Ideal}} + \dot{m}_{\mathrm{Leak}},$$

where:

- $\dot{m}$ is the actual mass flow rate.
- $\dot{m}_{\mathrm{Ideal}}$ is the ideal mass flow rate.
- $\dot{m}_{\mathrm{Leak}}$ is the internal leakage mas flow rate.

The torque generated at the motor is

$$\tau = \tau_{\mathrm{Ideal}} - \tau_{\mathrm{Friction}},$$

where:

- τ is the actual torque.
- τ_{Ideal} is the ideal torque.
- τ_{Friction} is the friction torque.

The ideal mass flow rate is

$$\dot{m}_{\mathrm{Ideal}} = \rho D \omega,$$

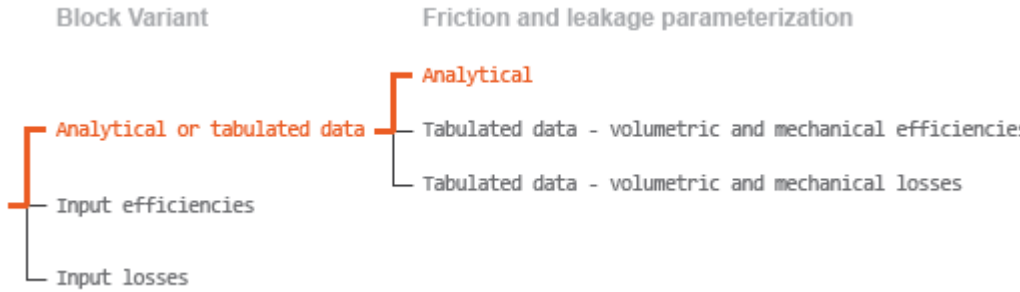
and the ideal generated torque is

$$\tau_{\text{Ideal}} = D\Delta p,$$

where:

- ρ is the average of the fluid densities at thermal liquid ports **A** and **B**.
- D is the **Displacement** parameter.
- ω is the shaft angular velocity.
- Δp is the pressure drop from inlet to outlet.

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$\dot{m}_{\text{Leak}} = \frac{K_{\text{HP}}\rho_{\text{Avg}}\Delta p}{\mu_{\text{Avg}}},$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}} \left| \Delta p \right| \tanh \frac{4\omega}{(5e-5)\,\omega_{\text{Nom}}} \right),$$

where:

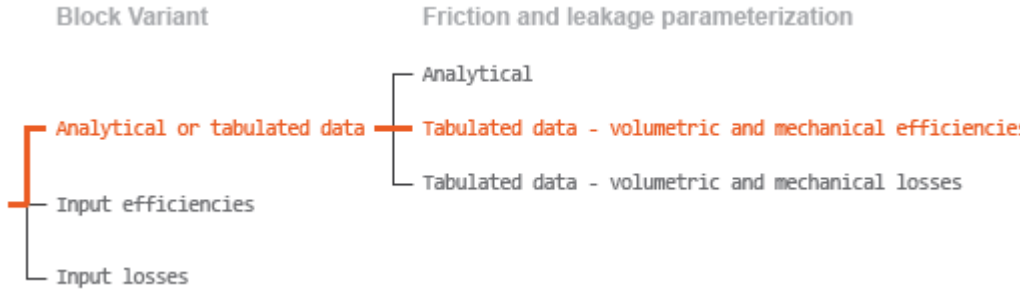
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- μ is the dynamic viscosity of the thermal liquid, taken here as the average of its values at the thermal liquid ports.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{\text{HP}} = \frac{D\omega_{\text{Nom}}\mu_{\text{Nom}} \left(\dfrac{1}{\eta_{\text{v,Nom}}} - 1 \right)}{\Delta p_{\text{Nom}}},$$

where:

- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- μ_{Nom} is the specified value of the **Nominal Dynamic viscosity** block parameter. This is the dynamic viscosity at which the nominal volumetric efficiency is specified.
- Δp_{Nom} is the specified value of the **Nominal pressure drop** block parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{\text{v,Nom}}$ is the specified value of the **Volumetric efficiency at nominal conditions** block parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`, the leakage flow rate is

$$\dot{m}_{\text{Leak}} = \dot{m}_{\text{Leak,Motor}} \frac{(1+\alpha)}{2} + \dot{m}_{\text{Leak,Pump}} \frac{(1-\alpha)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Motor}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Pump}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the motor-pump transition.
- $\dot{m}_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $\dot{m}_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure drop threshold for motor-pump transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.

The leakage flow rate is calculated from the volumetric efficiency, a quantity that is specified in tabulated form over the Δp–ω domain via the **Volumetric efficiency table** block parameter. When operating in motor mode (quadrants **1** and **3** of the Δp–ω chart shown in the "Operation Modes" on page 1-542 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak,Motor}} = \left(1-\eta_{\text{v}}\right)\dot{m},$$

where η_{v} is the volumetric efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in pump mode (quadrants **2** and **4** of the Δp–ω chart), the leakage flow rate is:

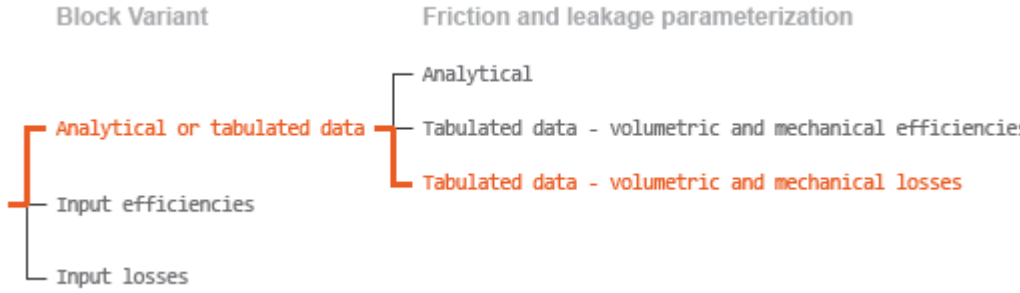
$$\dot{m}_{\text{Leak,Pump}} = -\left(1-\eta_{\text{v}}\right)\dot{m}_{\text{Ideal}}.$$

The friction torque is similarly calculated from the mechanical efficiency, a quantity that is specified in tabulated form over the Δp–ω domain via the **Mechanical efficiency table** block parameter. When operating in motor mode (quadrants **1** and **3** of the Δp–ω chart):

$$\tau_{\text{Friction,Motor}} = \left(1-\eta_{\text{m}}\right)\tau_{\text{Ideal}},$$

where η_{m} is the mechanical efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in pump mode (quadrants **2** and **4** of the Δp–ω chart):

$$\tau_{\text{Friction,Pump}} = -\left(1-\eta_{\text{m}}\right)\tau.$$



Case 3: Loss Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`, the leakage (volumetric) flow rate is specified directly in tabulated form over the Δp–ω domain:

$$q_{\text{Leak}} = q_{\text{Leak}}\left(\Delta p, \omega\right).$$

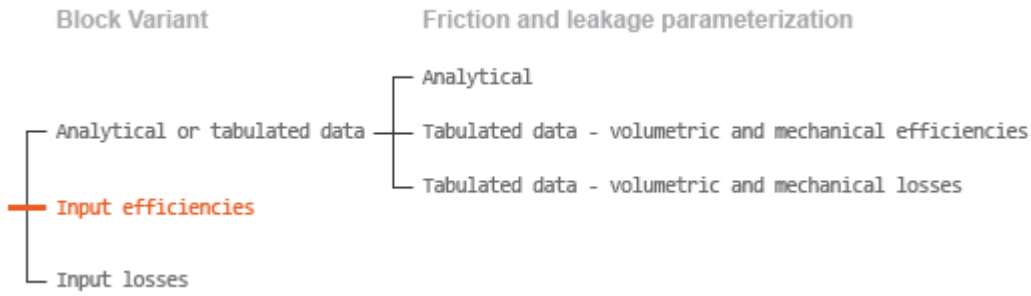
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega\right),$$

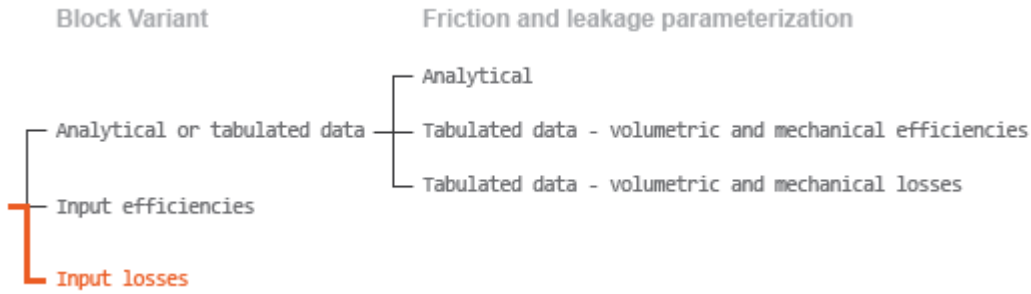
where $q_{\text{Leak}}(\Delta p,\omega)$ and $\tau_{\text{Friction}}(\Delta p,\omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.

The efficiencies are defined as positive quantities with value between zero and one. Input values outside of these bounds are set equal to the nearest bound (zero for inputs smaller than zero, one for inputs greater than one). In other words, the efficiency signals are *saturated* at zero and one.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

The signs of the inputs are ignored. The block sets the signs automatically from the operating conditions established during simulation—more precisely, from the Δp–ω quadrant in which the component happens to be operating. In other words, whether an input is positive or negative is irrelevant to the block.

Assumptions and Limitations

- The motor is treated as a quasi-steady component.
- The effects of fluid inertia and elevation are ignored.
- The motor wall is rigid.
- External leakage is ignored.

Ports

Input

`EV` — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

`EM` — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

LV — Volumetric loss, m^3/s

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the motor inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

LM — Mechanical loss, N*m

physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotating motor shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

A — Motor inlet

thermal liquid

Thermal liquid conserving port representing the motor inlet.

B — Motor outlet

thermal liquid

Thermal liquid conserving port representing the motor outlet.

C — Motor Case

mechanical rotational

Mechanical rotational conserving port representing the motor case.

R — Motor Shaft

mechanical rotational

Mechanical rotational conserving port representing the rotational motor shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-1305.

Variant 1: Analytical or tabulated data

`Displacement` — Fluid volume displaced per unit shaft rotation angle
`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque
`Analytical` (default) | `Tabulated data – volumetric and mechanical efficiencies` | `Tabulated data – volumetric and mechanical losses`

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup tables to map pressure drop, angular velocity, and displacement to component efficiencies or losses.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the volumetric efficiency
`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotary shaft at which the component’s volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer’s data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure drop` — Pressure drop at which to specify the volumetric efficiency
`10 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal dynamic viscosity` — Dynamic viscosity at which to specify the volumetric efficiency

`0.9 cP` (default) | scalar with units of area/time

Dynamic viscosity of the hydraulic fluid at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions

`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation

`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure drop coefficient at nominal displacement` — Proportionality constant at nominal displacement between friction torque and pressure drop

`0.6 N*m/MPa` (default) | scalar with units of torque/pressure

Proportionality constant at nominal displacement between the friction torque on the mechanical shaft and the pressure drop from inlet to outlet.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet

`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Pressure drop vector for efficiencies, dp` — Pressure drops at which to specify the volumetric and mechanical efficiencies

M-element vector with units of pressure (default)

M-element vector of pressure drops at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies

N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Volumetric efficiency table, e_v(dp,w)` — Volumetric efficiencies at the specified pressure drops and angular velocities

unitless *M*-by-*N* matrix

M-by-*N* matrix with the volumetric efficiencies at the specified fluid pressure drops, shaft angular velocities, and displacements. The efficiencies must be in the range of `0`–`1`. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w)` — Mechanical efficiencies at the specified pressure drops and angular velocities

unitless *M*-by-*N* matrix

M-by-*N* matrix with the mechanical efficiencies corresponding to the specified fluid pressure drops and shaft angular velocities. The efficiencies must be in the range of `0`–`1`. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes

`1e-3 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Angular velocity threshold for motor-pump transition` — Shaft angular velocity at which to initiate a smooth transition between pump and motor modes

`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Check if operating beyond the octants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure drop, shaft angular velocity, or instantaneous displacement cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies` or `Tabulated data – volumetric and mechanical losses`.

`Pressure drop vector for losses, dp` — Pressure drops at which to specify the volumetric and mechanical losses
M-element vector with units of pressure (default)

M-element vector of pressure drops at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses
N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

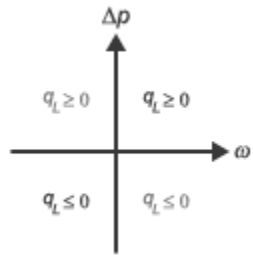
`Volumetric loss table, q_loss(dp,w)` — Internal leakage flow rates at the specified pressure drops and angular velocities

M-by-*N* matrix with units of volume/time

M-by-*N* matrix with the volumetric losses at the specified fluid pressure drops and shaft angular velocities. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants. The tabulated data for the volumetric losses must obey the convention shown in the figure, with positive values at positive pressure drops and negative values at negative pressure drops.



Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

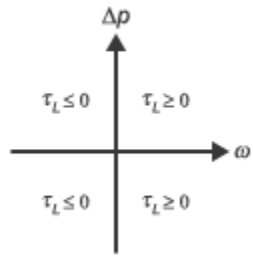
`Mechanical loss table, torque_loss(dp,w)` — Friction torques at the specified pressure drops and angular velocities

M-by-*N* matrix with units of torque

M-by-*N* matrix with the mechanical losses at the specified fluid pressure drops and shaft angular velocities. Mechanical loss is defined here as the friction torque due to seals and internal components. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants. The tabulated data for the mechanical losses must obey the convention shown in the figure, with positive values at positive angular velocities and negative values at negative angular velocities.



Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Displacement` — Fluid volume displaced per unit shaft rotation angle

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal
`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at this value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for motor-pump transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

Variant 3: Input losses

`Displacement` — Fluid volume displaced per unit shaft rotation angle
`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Angular velocity threshold for motor-pump transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Check if operating beyond the motor mode` — Simulation warning mode for operating conditions outside motoring mode
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the motoring mode. A warning is issued if the motor transitions to pumping mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Variables

`Mass flow rate into port A` — Mass flow rate into the component through thermal liquid port A

`1 kg/s` (default) | scalar with units of mass/time

Mass of fluid entering the component through the inlet per unit time at the start of simulation.

See Also

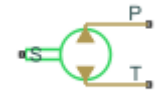
Fixed-Displacement Pump (TL)

Introduced in R2016a

Fixed-Displacement Pump

Mechanical-to-hydraulic power conversion device

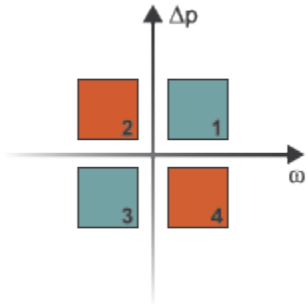
Library: Simscape / Fluids / Hydraulics (Isothermal) / Pumps and Motors



Description

The Fixed-Displacement Pump block represents a device that extracts power from a mechanical rotational network and delivers it to a hydraulic (isothermal liquid) network. The pump displacement is fixed at a constant value that you specify through the **Displacement** parameter.

Ports **T** and **P** represent the pump inlets. Port **S** represents the pump drive shaft. During normal operation, the pressure gain from port **T** to port **P** is positive if the angular velocity at port **S** is positive also. This operation mode is referred to here as *forward pump*.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure gain from port **T** to port **P** (Δp) and on the angular velocity at port **S** (ω). The “Operation

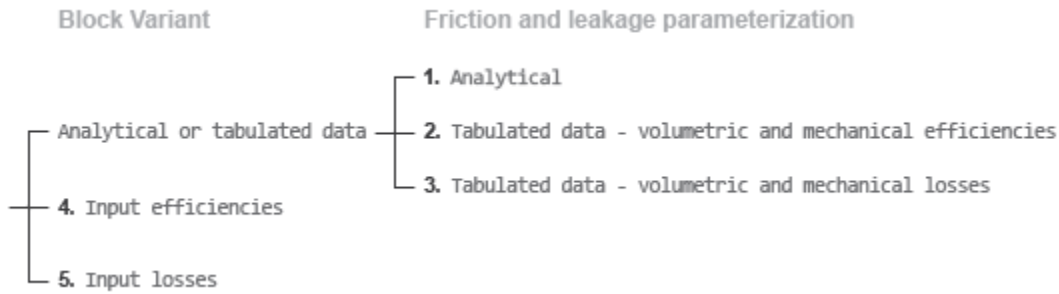
Modes” on page 1-587 figure maps the modes to the quadrants of a Δ*p*-*ω* chart. The modes are labeled 1–4:

- Mode **1**: forward pump — A positive shaft angular velocity generates a positive pressure gain.
- Mode **2**: reverse motor — A negative pressure drop (shown in the figure as a positive pressure gain) generates a negative shaft angular velocity.
- Mode **3**: reverse pump — A negative shaft angular velocity generates a negative pressure gain.
- Mode **4**: forward motor — A positive pressure drop (shown in the figure as a negative pressure gain) generates a positive shaft angular velocity.

The response time of the pump is considered negligible in comparison with the system response time. The pump is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The pump model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the pump inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Flow Rate and Driving Torque

The volumetric flow rate generated at the pump is

$$q = q_{\text{Ideal}} + q_{\text{Leak}},$$

where:

- q is the net volumetric flow rate.
- q_{Ideal} is the ideal volumetric flow rate.
- q_{Leak} is the internal leakage volumetric flow rate.

The driving torque required to power the pump is

$$\tau = \tau_{\text{Ideal}} + \tau_{\text{Friction}},$$

where:

- τ is the net driving torque.
- τ_{Ideal} is the ideal driving torque.
- τ_{Friction} is the friction torque.

The ideal volumetric flow rate is

$$q_{\text{Ideal}} = D\omega,$$

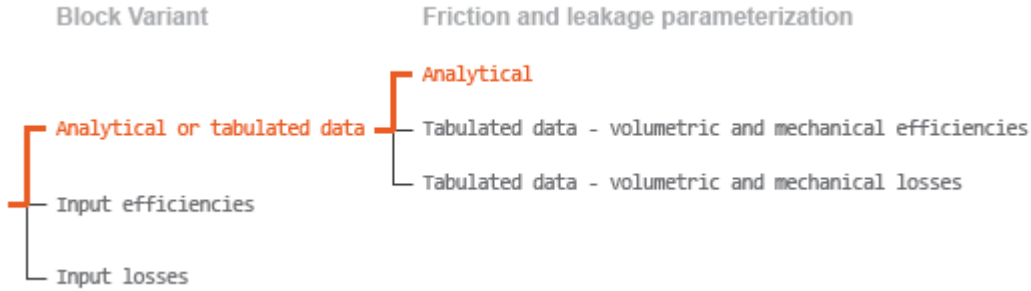
and the ideal driving torque is

$$\tau_{\text{Ideal}} = D\Delta p,$$

where:

- D is the specified value of the **Displacement** block parameter.
- ω is the instantaneous angular velocity of the rotary shaft.
- Δp is the instantaneous pressure gain from inlet to outlet.

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$q_{\text{Leak}} = K_{\text{HP}}\Delta p,$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}}|\Delta p|\right)\tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

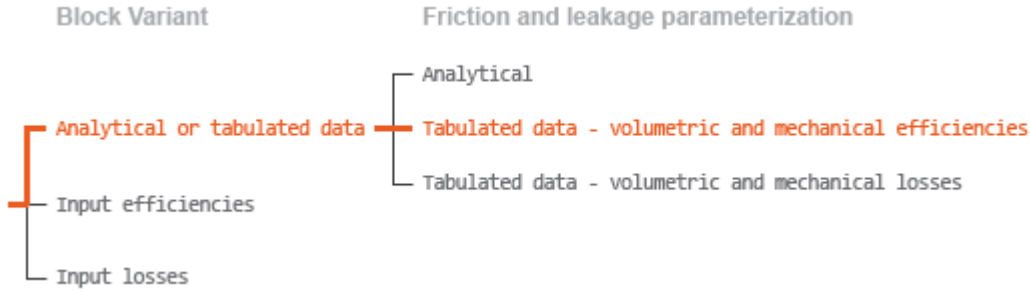
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure gain coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- $\omega_{Threshold}$ is the threshold angular velocity for the motor-pump transition. The threshold angular velocity is an internally set fraction of the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{HP} = \frac{\nu_{Nom}}{\rho \upsilon} \frac{\rho_{Nom} \omega_{Nom} D_{Max}}{\Delta p_{Nom}} \left(1 - \eta_{v,Nom}\right),$$

where:

- ν_{Nom} is the specified value of the **Nominal kinematic viscosity** block parameter. This is the kinematic viscosity at which the nominal volumetric efficiency is specified.
- ρ_{Nom} is the specified value of the **Nominal fluid density** block parameter. This is the density at which the nominal volumetric efficiency is specified.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- ρ is the actual fluid density in the attached hydraulic (isothermal liquid) network. This density can differ from the specified value of the **Nominal fluid density** block parameter.
- ν is the kinematic viscosity of the fluid associated with the fluid network.
- Δp_{Nom} is the specified value of the **Nominal pressure gain** block parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{v,Nom}$ is the specified value of the **Volumetric efficiency at nominal conditions** block parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`, the leakage flow rate is

$$q_{\text{Leak}} = q_{\text{Leak,Pump}} \frac{(1+\alpha)}{2} + q_{\text{Leak,Motor}} \frac{(1-\alpha)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Pump}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Motor}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the pump-pump transition.
- $q_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $q_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure drop threshold for motor-pump transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.

The leakage flow rate is computed from efficiency tabulated data through the equation

$$q_{\text{Leak,Pump}} = \left(1 - \eta_{\text{v}}\right) q_{\text{Ideal}},$$

in pump mode and through the equation

$$q_{\text{Leak,Motor}} = -\left(1 - \eta_{\text{v}}\right) q,$$

in motor mode, where:

- η_{v} is the volumetric efficiency obtained through interpolation or extrapolation of the **Volumetric efficiency table, e_v(dp,w)** parameter data.

Similarly, the friction torque is computed from efficiency tabulated data through the equation

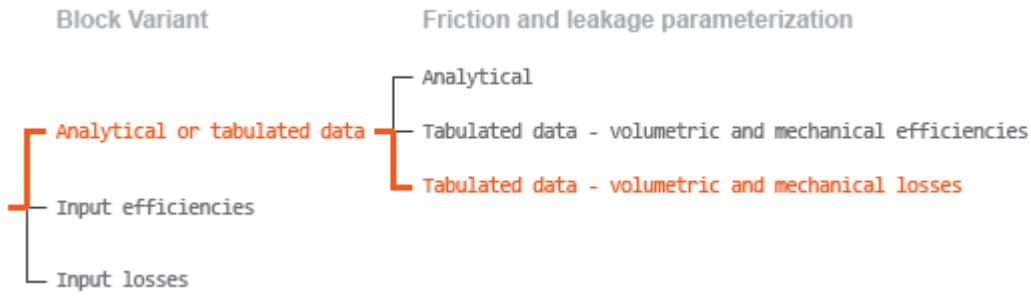
$$\tau_{\text{Friction,Pump}} = \left(1 - \eta_{\text{m}}\right) \tau,$$

in pump mode and through the equation

$$\tau_{\text{Friction,Motor}} = -\left(1 - \eta_{\text{m}}\right) \tau_{\text{Ideal}},$$

in motor mode, where:

- η_{m} is the mechanical efficiency obtained through interpolation or extrapolation of the **Mechanical efficiency table, e_m(dp,w)** parameter data.



Case 3: Loss Tabulated Data

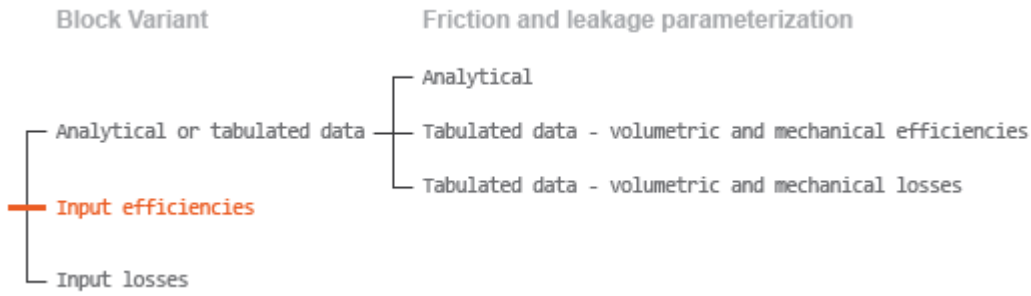
If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}\left(\Delta p, \omega\right).$$

and the friction torque equation is

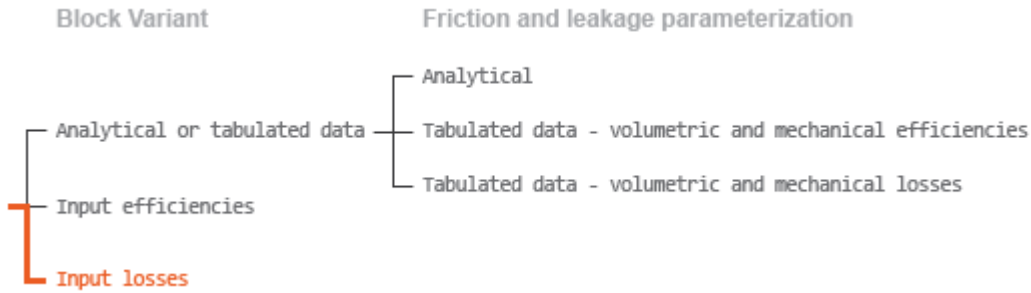
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega\right),$$

where $q_{\text{Leak}}(\Delta p,\omega)$ and $\tau_{\text{Friction}}(\Delta p,\omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the **Volumetric loss table, q_loss(dp,w)** and **Mechanical loss table, torque_loss (dp,w)** parameter data.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

Characteristic Curve Visualization

If the block variant is set to `Analytical or tabulated data`, you can plot a variety of performance, efficiency, and loss curves from simulation data and component parameters. Use the context-sensitive menu of the block to plot the characteristic curves. Right-click the block to open the menu and select **Fluids > Plot characteristic**. A test harness opens with instructions on how to generate the curves. See “Pump and Motor Characteristic Curves”.

Assumptions

- Fluid compressibility is negligible.
- Loading on the pump shaft due to inertia, friction, and spring forces is negligible.

Ports

Input

EV — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

EM — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

LV — Volumetric loss, `m^3/s`

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the pump inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

LM — Mechanical loss, `N*m`

physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotating pump shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

P — Pump outlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the pump outlet.

T — Pump inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the pump inlet.

S — Pump shaft
mechanical rotational

Mechanical rotational conserving port representing the pump shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-588.

Variant 1: Analytical or tabulated data

`Displacement` — Fluid volume displaced per unit shaft rotation angle
`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque
`Analytical` (default)

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup

tables to map pressure drop and angular velocity to component efficiencies or losses. The tabular options include:

- `Tabulated data — volumetric and mechanical efficiencies`
- `Tabulated data — volumetric and mechanical losses`

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data`.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the nominal volumetric efficiency

`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotating shaft at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure gain` — Pressure gain at which to specify the nominal volumetric efficiency

`100e5 Pa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal kinematic viscosity` — Kinematic viscosity at which to specify the nominal volumetric efficiency

`18 cSt` (default) | scalar with units of area/time

Kinematic viscosity of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal fluid density` — Fluid density at which to specify the nominal volumetric efficiency

`900 kg/m^3` (default) | scalar with units of mass/volume

Mass density of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions

`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation
`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the smoothed load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure gain coefficient` — Proportionality constant between friction torque and pressure gain
`0.6e-6 N*m/Pa` (default) | scalar with units of torque/pressure

Proportionality constant between the friction torque on the mechanical shaft and the pressure gain from inlet to outlet.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

`Pressure gain vector for efficiencies, dp` — Pressure gains at which to specify the volumetric and mechanical efficiencies
M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies
N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Volumetric efficiency table, e_v(dp,w)` — Volumetric efficiencies at the specified pressure gains and angular velocities
unitless *M*-by-*N* matrix

M-by-*N* matrix with the volumetric efficiencies at the specified fluid pressure gains and shaft angular velocities. The efficiencies must fall in the range of 0–1. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w)` — Mechanical efficiencies at the specified pressure gains and angular velocities

unitless *M*-by-*N* matrix

M-by-*N* matrix with the mechanical efficiencies corresponding to the specified fluid pressure gains and shaft angular velocities. The efficiencies must fall in the range of 0–1. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Pressure gain threshold for pump–motor transition` — Pressure gain at which to initiate a smooth transition between pumping and motoring modes

`1e5 Pa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Angular velocity threshold for pump–motor transition` — Shaft angular velocity at which to initiate a smooth transition between pumping and motoring modes

`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Check if operating beyond the quadrants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data

`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure gain or shaft angular velocity cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies` or `Tabulated data – volumetric and mechanical losses`.

`Pressure gain vector for losses, dp` — Pressure gains at which to specify the volumetric and mechanical losses

M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses

N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the component loss data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Volumetric loss table, q_loss(dp,w)` — Internal leakage flow rates at the specified pressure gains and angular velocities

M-by-*N* matrix with units of volume/time

M-by-*N* matrix with the volumetric gains at the specified fluid pressure gains and shaft angular velocities. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Mechanical loss table, torque_loss(dp,w)` — Friction torques at the specified pressure gains and angular velocities

M-by-*N* matrix with units of torque

M-by-*N* matrix with the mechanical losses at the specified fluid pressure gains and shaft angular velocities. Mechanical loss is defined here as the friction torque due to seals and internal components. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Pump Displacement` — Fluid volume displaced per unit shaft rotation angle
`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Pressure gain threshold for pump–motor transition` — Pressure gain at which to initiate a smooth transition between motor and pump modes
`1e5 Pa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for pump–motor transition` — Shaft angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure

`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning

`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

Variant 3: Input losses

`Pump Displacement` — Fluid volume displaced per unit shaft rotation angle

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Check if operating beyond the pump mode` — Simulation warning mode for operating conditions outside pumping mode

`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the pumping mode. A warning is issued if the pump transitions to motoring mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Input losses`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

See Also

Fixed-Displacement Motor | Fixed-Displacement Motor (TL) | Fixed-Displacement Pump (TL) | Variable-Displacement Motor | Variable-Displacement Pump

Introduced in R2006b

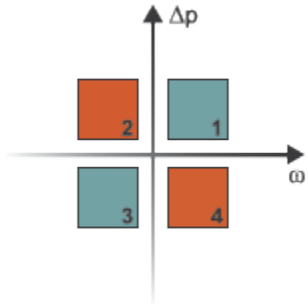
Fixed-Displacement Pump (TL)

Mechanical-hydraulic power conversion device
Library: Simscape / Fluids / Thermal Liquid / Pumps & Motors

Description

The Fixed-Displacement Pump (TL) block represents a device that extracts power from a mechanical rotational network and delivers it to a hydraulic (isothermal liquid) network. The pump displacement is fixed at a constant value that you specify through the **Displacement** parameter.

Ports **A** and **B** represent the pump inlets. Ports **R** and **C** represent the motor drive shaft and case. During normal operation, the pressure gain from port **A** to port **B** is positive if the angular velocity at port **R** relative to port **C** is positive also. This operation mode is referred to here as *forward pump*.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure gain from port **A** to port **B** (Δp) and on the angular velocity at port **R** relative to port **C**

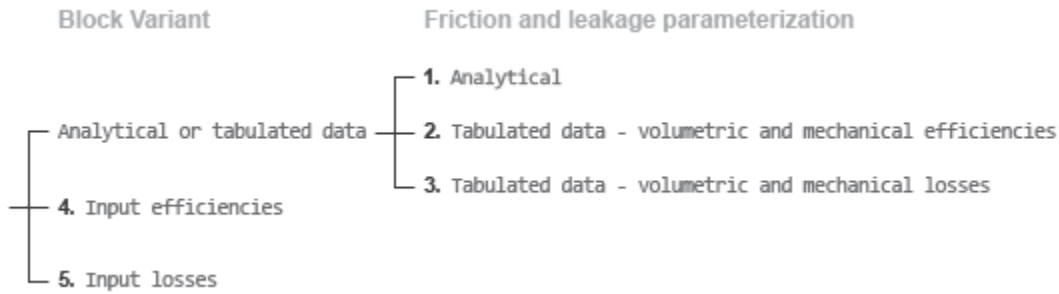
(ω). The “Operation Modes” on page 1-587 figure maps the modes to the quadrants of a Δp-ω chart. The modes are labeled 1–4:

- Mode **1**: forward pump — A positive shaft angular velocity generates a positive pressure gain.
- Mode **2**: reverse motor — A negative pressure drop (shown in the figure as a positive pressure gain) generates a negative shaft angular velocity.
- Mode **3**: reverse pump — A negative shaft angular velocity generates a negative pressure gain.
- Mode **4**: forward motor — A positive pressure drop (shown in the figure as a negative pressure gain) generates a positive shaft angular velocity.

The response time of the pump is considered negligible in comparison with the system response time. The pump is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The pump model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the pump inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Flow Rate and Driving Torque

The mass flow rate generated at the pump is

$$\dot{m} = \dot{m}_{\mathrm{Ideal}} - \dot{m}_{\mathrm{Leak}},$$

where:

- $\dot{m}$ is the actual mass flow rate.
- $\dot{m}_{\mathrm{Ideal}}$ is the ideal mass flow rate.
- $\dot{m}_{\mathrm{Leak}}$ is the internal leakage mas flow rate.

The driving torque required to power the pump is

$$\tau = \tau_{\mathrm{Ideal}} + \tau_{\mathrm{Friction}},$$

where:

- τ is the actual driving torque.
- τ_{Ideal} is the ideal driving torque.
- τ_{Friction} is the friction torque.

The ideal mass flow rate is

$$\dot{m}_{\mathrm{Ideal}} = \rho D \omega,$$

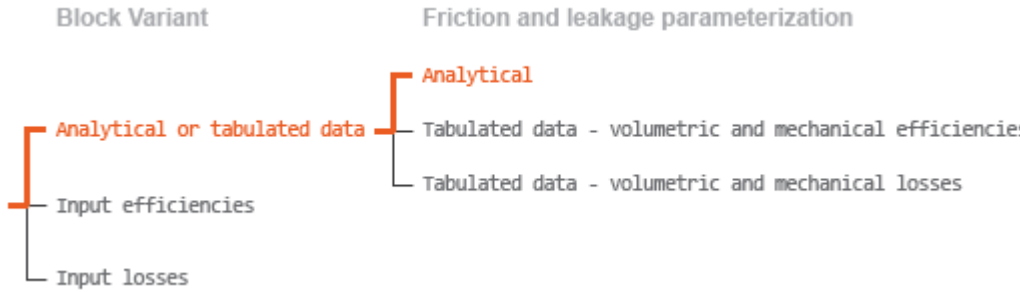
and the ideal generated torque is

$$\tau_{\text{Ideal}} = D\Delta p,$$

where:

- ρ is the average of the fluid densities at thermal liquid ports **A** and **B**.
- D is the **Displacement** parameter.
- ω is the shaft angular velocity.
- Δp is the pressure drop from inlet to outlet.

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$\dot{m}_{\text{Leak}} = \frac{K_{\text{HP}}\rho_{\text{Avg}}\Delta p}{\mu_{\text{Avg}}},$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}} |\Delta p| \tanh \frac{4\omega}{(5e-5)\,\omega_{\text{Nom}}} \right),$$

where:

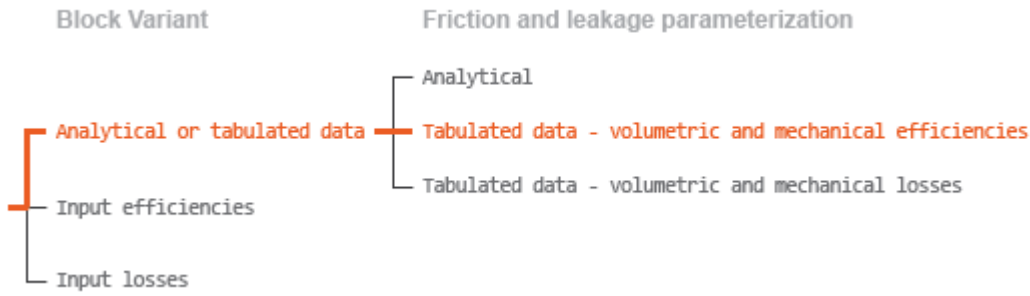
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- μ is the dynamic viscosity of the thermal liquid, taken here as the average of its values at the thermal liquid ports.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{\text{HP}} = \frac{D\omega_{\text{Nom}}\mu_{\text{Nom}}\left(1-\eta_{\text{v,Nom}}\right)}{\Delta p_{\text{Nom}}},$$

where:

- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- μ_{Nom} is the specified value of the **Nominal Dynamic viscosity** block parameter. This is the dynamic viscosity at which the nominal volumetric efficiency is specified.
- Δp_{Nom} is the specified value of the **Nominal pressure drop** block parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{\text{v,Nom}}$ is the specified value of the **Volumetric efficiency at nominal conditions** block parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`, the leakage flow rate is

$$\dot{m}_{\text{Leak}} = \dot{m}_{\text{Leak,Pump}} \frac{(1+\alpha)}{2} + \dot{m}_{\text{Leak,Motor}} \frac{(1-\alpha)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Pump}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Motor}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the motor-pump transition.
- $\dot{m}_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $\dot{m}_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure gain threshold for pump-motor transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for pump-motor transition** block parameter.

The leakage flow rate is calculated from the volumetric efficiency, a quantity that is specified in tabulated form over the Δp–ω domain via the **Volumetric efficiency table** block parameter. When operating in pump mode (quadrants **1** and **3** of the Δp–ω chart shown in the "Operation Modes" on page 1-542 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak,Pump}} = \left(1-\eta_{\text{v}}\right)\dot{m}_{\text{Ideal}},$$

where η_{v} is the volumetric efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in motor mode (quadrants **2** and **4** of the Δp–ω chart), the leakage flow rate is:

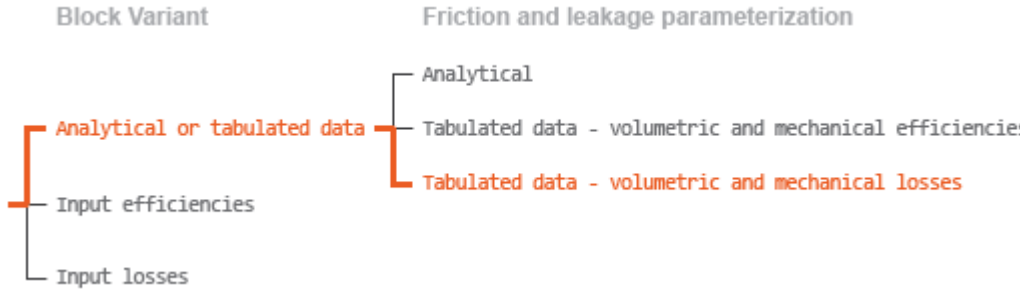
$$\dot{m}_{\text{Leak,Motor}} = -\left(1-\eta_{\text{v}}\right)\dot{m}.$$

The friction torque is similarly calculated from the mechanical efficiency, a quantity that is specified in tabulated form over the Δp–ω domain via the **Mechanical efficiency table** block parameter. When operating in pump mode (quadrants **1** and **3** of the Δp–ω chart):

$$\tau_{\text{Friction,Pump}} = \left(1-\eta_{\text{m}}\right)\tau,$$

where η_{m} is the mechanical efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in motor mode (quadrants **2** and **4** of the Δp–ω chart):

$$\tau_{\text{Friction,Motor}} = -\left(1-\eta_{\text{m}}\right)\tau_{\text{Ideal}}.$$



Case 3: Loss Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`, the leakage (volumetric) flow rate is specified directly in tabulated form over the Δp–ω domain:

$$q_{\text{Leak}} = q_{\text{Leak}}\left(\Delta p, \omega\right).$$

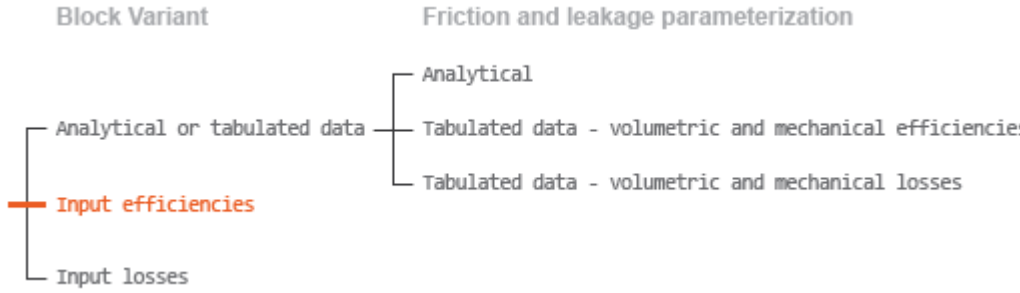
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega\right),$$

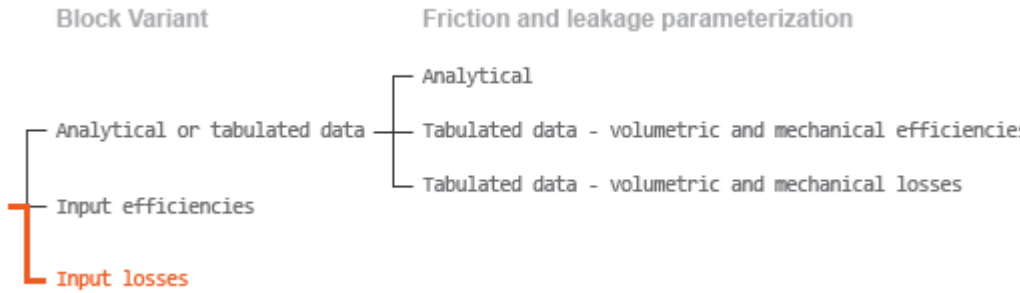
where $q_{\text{Leak}}(\Delta p,\omega)$ and $\tau_{\text{Friction}}(\Delta p,\omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.

The efficiencies are defined as positive quantities with value between zero and one. Input values outside of these bounds are set equal to the nearest bound (zero for inputs smaller than zero, one for inputs greater than one). In other words, the efficiency signals are *saturated* at zero and one.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

The signs of the inputs are ignored. The block sets the signs automatically from the operating conditions established during simulation—more precisely, from the Δp–ω quadrant in which the component happens to be operating. In other words, whether an input is positive or negative is irrelevant to the block.

Assumptions and Limitations

- The pump is treated as a quasi-steady component.
- The effects of fluid inertia and elevation are ignored.
- The pump wall is rigid.
- External leakage is ignored.

Ports

Input

`EV` — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

`EM` — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

`LV` — Volumetric loss, `m^3/s`

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the pump inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

`LM` — Mechanical loss, `N*m`

physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotary pump shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

`A` — Pump inlet

thermal liquid

Thermal liquid conserving port representing the pump inlet.

`B` — Pump outlet

thermal liquid

Thermal liquid conserving port representing the pump outlet.

`C` — Pump case

mechanical rotational

Mechanical rotational conserving port representing the pump case.

`R` — Pump shaft

mechanical rotational

Mechanical rotational conserving port representing the rotary pump shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-1365.

Variant 1: Analytical or tabulated data

`Displacement` — Fluid volume displaced per unit shaft rotation angle

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque

`Analytical` (default) | `Tabulated data – volumetric and mechanical efficiencies` | `Tabulated data – volumetric and mechanical losses`

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup tables to map pressure drop, angular velocity, and displacement to component efficiencies or losses.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the volumetric efficiency

`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotary shaft at which the component’s volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer’s data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure gain` — Pressure gain at which to specify the volumetric efficiency

`10 MPa` (default)

Pressure drop from inlet to outlet at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal dynamic viscosity` — Dynamic viscosity at which to specify the volumetric efficiency

`0.9 cP` (default) | scalar with units of area/time

Dynamic viscosity of the hydraulic fluid at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions

`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation

`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure gain coefficient at nominal displacement` — Proportionality constant at maximum displacement between friction torque and pressure gain

`0.6 N*m/MPa` (default) | scalar with units of torque/pressure

Proportionality constant at maximum displacement between the friction torque on the mechanical shaft and the pressure gain from inlet to outlet.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet

`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Pressure gain vector for efficiencies, dp` — Pressure gains at which to specify the volumetric and mechanical efficiencies

M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies

N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Volumetric efficiency table, e_v(dp,w)` — Volumetric efficiencies at the specified pressure gains and angular velocities

unitless *M*-by-*N* matrix

M-by-*N* matrix with the volumetric efficiencies at the specified fluid pressure gains and shaft angular velocities. The efficiencies must be in the range of `0`–`1`. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w)` — Mechanical efficiencies at the specified pressure gains and angular velocities

unitless *M*-by-*N* matrix

M-by-*N* matrix with the mechanical efficiencies corresponding to the specified fluid pressure gains and shaft angular velocities. The efficiencies must be in the range of `0`–`1`. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Pressure gain threshold for pump-motor transition` — Pressure gain at which to initiate a smooth transition between pump and motor modes

`1e-3 MPa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Angular velocity threshold for pump-motor transition` — Shaft angular velocity at which to initiate a smooth transition between pump and motor modes

`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Check if operating beyond the octants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure gain, shaft angular velocity, or instantaneous displacement cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies` or `Tabulated data – volumetric and mechanical losses`.

`Pressure gain vector for losses, dp` — Pressure gains at which to specify the volumetric and mechanical losses
M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses
N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

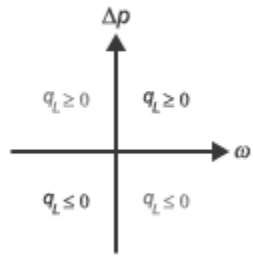
`Volumetric loss table, q_loss(dp,w)` — Internal leakage flow rates at the specified pressure gains and angular velocities

M-by-*N* matrix with units of volume/time

M-by-*N* matrix with the volumetric losses at the specified fluid pressure gains and shaft angular velocities. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants. The tabulated data for the volumetric losses must obey the convention shown in the figure, with positive values at positive pressure gains and negative values at negative pressure gains.



Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

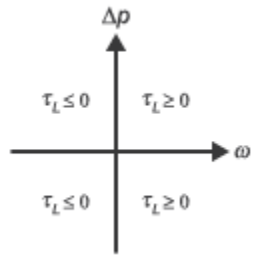
`Mechanical loss table, torque_loss(dp,w)` — Friction torques at the specified pressure gains and angular velocities

M-by-*N* matrix with units of torque

M-by-*N* matrix with the mechanical losses at the specified fluid pressure gains and shaft angular velocities. Mechanical loss is defined here as the friction torque due to seals and internal components. *M* and *N* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.

The tabulated data need not encompass all quadrants of operation—those of a (ω, Δp) chart. It suffices to specify the data for a single quadrant. Refer to the block description for the operation modes corresponding to the various quadrants. The tabulated data for the mechanical losses must obey the convention shown in the figure, with positive values at positive angular velocities and negative values at negative angular velocities.



Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Displacement` — Fluid volume displaced per unit shaft rotation angle

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal
`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at this value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Pressure gain threshold for pump-motor transition` — Pressure gain at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for motor-pump transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

Variant 3: Input losses

`Displacement` — Fluid volume displaced per unit shaft rotation angle
`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid volume displaced per unit shaft rotation angle. The displacement is fixed at this value during simulation. The specified value must be greater than zero.

`Pressure gain threshold for pump-motor transition` — Pressure gain at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Angular velocity threshold for pump-motor transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Check if operating beyond the motor mode` — Simulation warning mode for operating conditions outside motoring mode
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the motoring mode. A warning is issued if the motor transitions to pumping mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Variables

`Mass flow rate into port A` — Mass flow rate into the component through thermal liquid port A

`1 kg/s` (default) | scalar with units of mass/time

Mass of fluid entering the component through the inlet per unit time at the start of simulation.

See Also

Centrifugal Pump (TL) | Fixed-Displacement Motor (TL) | Variable-Displacement Pump (TL)

Introduced in R2016a

Flow Coefficient Parameterized Valve (TL)

Valve with flow area modeled in terms of its Cv or Kv flow coefficient
Library: Simscape / Fluids / Thermal Liquid / Valves & Orifices

Description

The Flow Coefficient Parameterized Valve (TL) block models the flow characteristics of a valve using the flow coefficient as a means to relate the flow rate through the ports to the pressure drop across them. The flow coefficient parameterization suits those cases in which the internal geometry of a valve is known in poor detail or in which the flow coefficient is the preferred means of characterizing a valve. The valve model is sufficiently general in its assumptions that it can be applied to a variety of valves; no single valve type is assumed in this block.

Flow Coefficient Parameterization

The flow coefficient measures the ease with which a fluid driven by a pressure differential flows across a valve. It is an important parameter in the sizing of valves and a convenient way of characterizing one in a model. In this role it is especially useful, as it quantifies in a single number all of the losses incurred in the flow passages of the valve. The flow coefficient is typically measured by the valve manufacturer and reported in a data sheet provided for the valve.

Strictly speaking there are two flow coefficients in common use: C_v and K_v. These describe what is fundamentally the same quantity: the volumetric flow rate of water passing through a valve that has been opened to its maximum and set to some standard operating conditions. The flow coefficients differ only in the temperature and pressure drop established during the measurement (the operating conditions) and in the physical units used in its expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Flow coefficient specification** block parameter is set to `Cv coefficient (USG/min)` [1 on page 1-640].

- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `bar`; it is expressed in metric units of m^3/h. This is the flow coefficient used in the model when the **Flow coefficient specification** block parameter is set to `Kv coefficient (m^3/h)`. [2 on page 1-640].

Two values are required of the flow coefficient, a *maximum* and a *minimum*. The maximum corresponds to a valve open to full capacity; this is the value frequently reported in valve data sheets. The minimum corresponds to a valve closed tight, when only leakage flow remains, if any at all. This lower bound imposed on the flow coefficient serves primarily to ensure the numerical robustness of the model. Its exact value is less important than its being a (generally very small) number greater than zero.

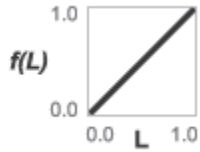
Valve Opening Parameterization

The opening fraction of the valve (the ratio of the valve opening area to its maximum value) is determined during simulation from the input specified at port **L**. This input is the *control signal* and it is, in some valves, associated with *stroke* or *lift percent*. The control signal can range in value from `0` to `1`. If a lesser or greater value is specified, it is adjusted to the nearest of the two limits. In other words, the signal is *saturated* at `0` and `1`.

The conversion from control signal to opening fraction depends on the **Valve opening characteristics** parameterization selected in the block. The calculation result is the same for all parameterizations when the control signal is either `0` or `1`: a value of `0` at port **L** always means a maximally (though not necessarily *completely*) closed valve; a value of `1` always means a maximally open valve. At intermediate values, the opening fractions differ:

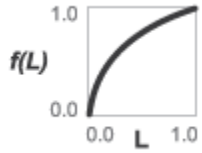
- `Linear` — The valve opening fraction (denoted $f(L)$ in the figure) is equal to the control signal at port **L**. The two vary in tandem until the control signal either drops below zero (the valve is fully closed) or rises above its maximum value (the valve is fully open). The opening fraction ranges from `0` to `1` (a statement that does not apply to all valve opening parameterizations).

  $$f(L) = L.$$



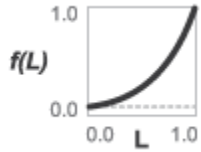
- `Quick opening` — The valve opening fraction, $f(L)$, is a power function of the control signal at port **L**. The opening fraction rises quickly with the control signal at values near `0` and slowly at values near `1`. The exponent (α) in the calculation determines how the opening rate changes with the control signal. The opening fraction ranges from `0` to `1`.

  $$f(L) = L^{1/\alpha}, \quad \alpha > 1.$$

- `Equal percentage` — The valve opening fraction is an exponential function of the control signal at port **L**. The opening fraction rises slowly with the control signal at values near `0` and quickly at values near `1`. The base of the exponent (R, the *rangeability* of the valve, the ratio between the maximum and minimum flow rates through the ports) determines how the opening rate changes with the control signal. The opening fraction ranges from a small fraction (`0.02`–`0.05` for typical rangeability values of `20`–`50`) to `1`.

  $$f(L) = R^{L-1}, \quad R \approx 20-50.$$

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of fluid can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through the port indicated by the subscript (**A** or **B**).

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. It is this cumulative effect that the flow coefficient of the valve enables the block to capture in a model. This constant factors in the momentum balance of the valve *indirectly*, by way of the effective opening area (S) of the same:

$$p_{\mathrm{A}} - p_{\mathrm{B}} = \frac{\dot{m}\sqrt{\dot{m}^2 + \dot{m}_{\mathrm{cr}}}}{\rho_{\mathrm{avg}} S^2},$$

where:

- p is the pressure at the port indicated by the subscript (**A** or **B**).
- ρ_{avg} is the arithmetic mean of the fluid densities at the ports.
- $\dot{m}$ is the mass flow rate at a port.
- $\dot{m}_{cr}$ is the critical mass flow rate, that at which the flow is assumed to switch between laminar and turbulent regimes:

  $$\dot{m}_{cr} = \mathrm{Re}_{\mathrm{cr}} \mu_{\mathrm{avg}} \sqrt{\frac{\pi}{4} S},$$

where Re_{cr} is the Reynolds number at the transition point and μ_{avg} is the average of the dynamic viscosities at the ports.

The effective opening area (S) is the product of scaling of the maximum opening area of the valve by the opening fraction ($f(L)$). The calculation imposes a lower bound on the opening area at which only leakage flow remains:

$$S(L) = \begin{cases} S_{\max} f(L), & \text{if } S_{\max} f(L) > S_{\min} \\ S_{\min}, & \text{else} \end{cases},$$

where the subscripts `max` and `min` refer to a valve in the fully opened and maximally closed positions; the function $f(L)$ is the opening fraction calculated from the control signal specified at port **L**:

$$f(L) = \begin{cases} L, & \text{Linear parameterization} \\ L^{1/\alpha}, & \text{Quick opening parameterization} \\ R^{L-1}, & \text{Equal percentages parameterization} \end{cases},$$

where α is obtained from the **Exponent number** block parameter and R from the **Rangeability** parameter. The maximum opening area is obtained by multiplying the (maximum) flow coefficient by an appropriate conversion factor:

$$S_{\text{Max}} = \begin{cases} \lambda_C C_{v,\max}, & \text{if } C_v \text{ is used} \\ \lambda_K K_{v,\max}, & \text{if } K_v \text{ is used} \end{cases},$$

where:

- λ_C is the conversion factor between the maximum flow coefficient in imperial units ($C_{v,max}$) and the maximum opening area: `2.4015e-5 m^2`.
- λ_K is the conversion factor between the maximum flow coefficient in SI units ($K_{v,max}$) and the maximum opening area: `2.7765e-5 m^2`.

The minimum opening area in turn is obatined by multiplying the maximum opening area by the ratio between minimum and maximum flow coefficients:

$$S_{\mathrm{Min}} = \begin{cases} \dfrac{C_{\mathrm{v,min}}}{C_{\mathrm{v,max}}} S_{\mathrm{max}}, & \text{if } C_{\mathrm{v}} \text{ is used} \\ \dfrac{K_{\mathrm{v,min}}}{K_{\mathrm{v,max}}} S_{\mathrm{max}}, & \text{if } K_{\mathrm{v}} \text{ is used} \end{cases},$$

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the fluid and the wall that surrounds it. No work is done on or by the fluid as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Input

`L` — Valve control signal, `unitless`

physical signal

Port through which to specify the signal that controls the opening of the valve (in certain valve types associated with *stroke* or *lift percent*). The valve is fully closed at a value of `0` and fully open at a value of `1`.

Conserving

`A` — Flow passage

thermal liquid

Opening through which the flow can enter or exit the flow resistance. Which of the ports serves as inlet and which as outlet depends on the direction of flow.

B — Flow passage
thermal liquid

Opening through which the flow can enter or exit the flow resistance. Which of the ports serves as inlet and which as outlet depends on the direction of flow.

Parameters

`Flow coefficient specification` — Parameterization by which to specify the flow coefficient of the valve
`Cv coefficient (USG/min)` (default) | `Kv coefficient (m^3/h)`

Choice of flow coefficient to specify. C_v and K_v differ in the physical conditions of their measurement and in the physical units of their expression. Generally at least one of these is reported in product data sheets provided by the manufacturer. Select the option for which data is available.

`Cv coefficient at maximum flow` — Flow coefficient expressed in imperial units for a maximally open valve
1 (default) | positive scalar in units of USG/min

Flow coefficient expressed in imperial units for a valve opened to full capacity. This is the value reported in the valve data sheets when the C_v coefficient is provided by the manufacturer.

`Kv coefficient at maximum flow` — Flow coefficient expressed in SI units for a maximally open valve
1 (default) | positive scalar in units of m^3/h

Flow coefficient expressed in SI units for a valve opened to full capacity. This is the value reported in the valve data sheets when the K_v coefficient is provided by the manufacturer.

`Cv coefficient at minimum flow` — Flow coefficient expressed in imperial units for a maximally closed valve
`1e-4` (default) | positive scalar in units of USG/min

Flow coefficient expressed in imperial units for a valve that is closed tight and devoid of all flow except for that due to internal leakage between port **A** and port **B**. The primary purpose of this parameter is to ensure the numerical robustness of the model during simulation. Its exact value is less important than its being a very small number greater than zero.

`Kv coefficient at minimum flow` — Flow coefficient expressed in SI units for a maximally closed valve
`1e-4` (default) | positive scalar in units of m^3/h

Flow coefficient expressed in SI units for a valve that is closed tight and devoid of all flow except for that due to internal leakage between port **A** and port **B**. The primary purpose of this parameter is to ensure the numerical robustness of the model during simulation. Its exact value is less important than it being a very small number greater than zero.

`Valve opening characteristics` — Parameterization by which to compute the valve opening fraction
`Linear` (default) | `Quick openingEqual percentages`

Choice of valve opening profile to use during simulation. The selection made here determines how the control signal specified at port **L** is converted into a valve opening fraction (and opening area). See the block description for detail on the opening parameterizations.

`Exponent number` — Denominator of the exponent in the quick-opening parameterization
2 (default) | positive unitless scalar

Number by which to divide the exponent in the governing equation of the quick-opening parameterization. Use a value greater than `1` for a true quick-opening profile, one corresponding to a valve that opens quickly from a maximally closed position but progressively more slowly as it approaches the maximally opened position.

`Valve rangeability` — Base of the exponent in the equal-percentages parameterization
`50` (default) | positive unitless scalar

Number to which the exponent is applied in the governing equation of the equal-percentages parameterization. This number is the ratio between the maximum and minimum flow rates through the valve (the first corresponding obtained when the valve is maximally opened and the second when the valve is maximally closed). Typical values range from `20` to `50`.

`Cross-sectional area at ports A and B` — Flow area at the inlet and outlet of the valve
`0.01 m` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the valve connects.

`Characteristic longitudinal length` — Measure of the length of the flow path through the valve
`0.1` (default) | positive scalar in units of length

Average distance traversed by the fluid as it travels from inlet to outlet. This distance is used in the calculation of the internal thermal conduction that occurs between the two ports (as part of the smoothed upwind energy scheme employed in the thermal liquid domain).

`Critical Reynolds number` — Reynolds number at which the flow switches between laminar and turbulent
12 (default) | positive unitless scalar

Reynolds number at which the fluid in the valve is expected to switch between its laminar and turbulent flow regimes. The switch is assumed to occur at once: the transition region that normally exists between the two flow regimes is ignored.

Variables Tab

`Mass flow rate into port A` — Desired initial value of the mass flow rate at port A
cleared (default) | checked

Mass flow rate to apply at port **A** at the start of simulation. The block uses this value as a guide when preparing the model for simulation. If the value is compatible with the constraints of the model, then it is used to configure the initial state of the valve. If the value conflicts with initial state variables specified in other blocks, then its **Priority** setting determines whether it is applied precisely, approximately, or not at all.

References

[1] *Control Valve Handbook*. 4th ed. Marshalltown, IA: Fisher Controls International. 2005.

[2] *Flow of Fluids through Valves, Fittings and Pipe*. Stamford, CT: Crane, 2010.

See Also

Ball Valve (TL) | Check Valve (TL) | Gate Valve (TL)

Introduced in R2016a

Flow Divider

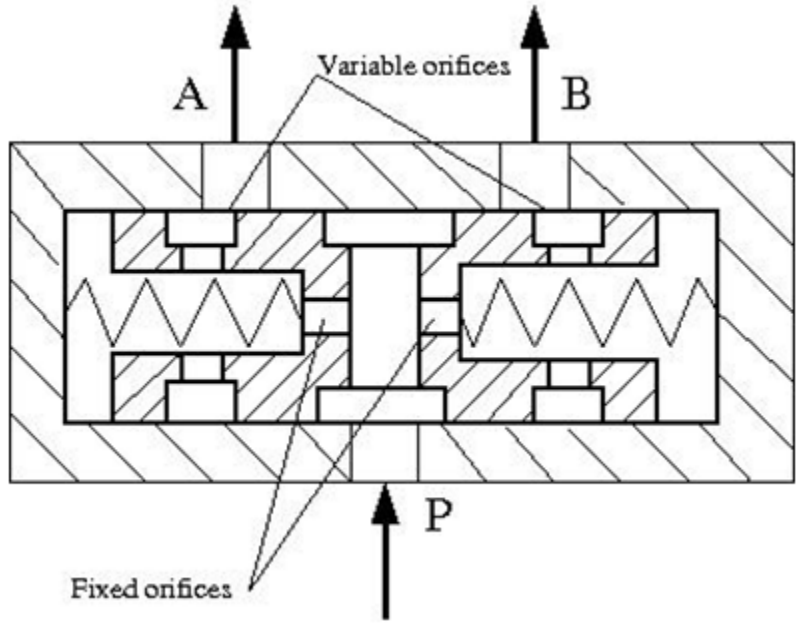
Hydraulic two-path flow divider

Library

Flow Control Valves

Description

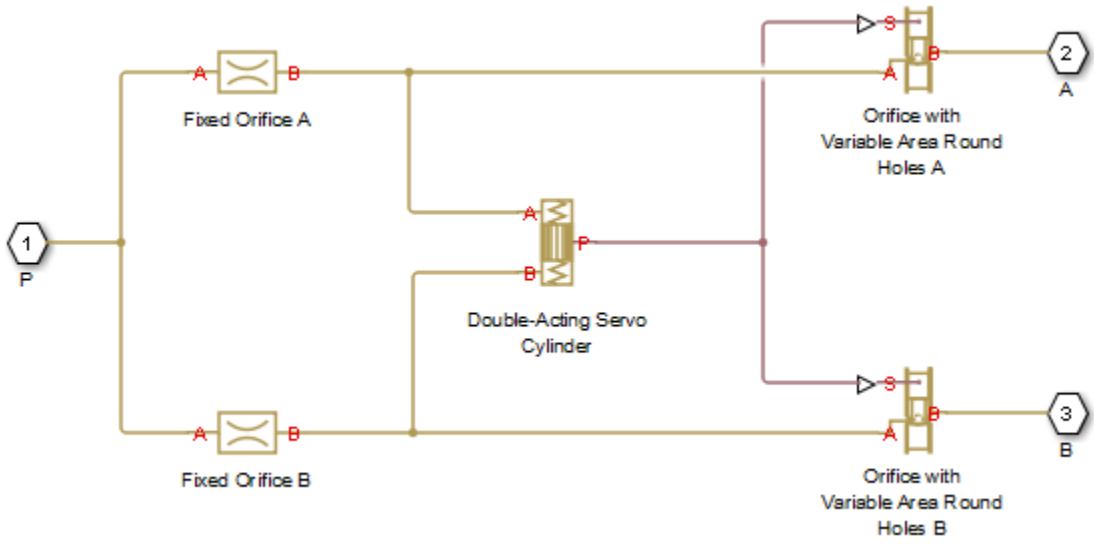
The Flow Divider block simulates a hydraulic two-path flow divider, which consists of a spring-centered spool installed in a case, as shown in the figure.



The flow from the source enters the valve through port P and is split into two parts flowing through the P–A and P–B paths. Each path contains a fixed orifice and a variable orifice. The fixed orifices must be precisely matched to divide flow in equal parts, or arranged in a certain proportion if unequal division is required.

The purpose of variable orifices is to maintain a constant pressure drop across the fixed orifices, regardless of pressure fluctuations at valve outlets. The load increase on any outlet causes the pressure drop across the spool (and across fixed orifices) to change, and thus shift the spool. As a result, the passage areas of variable orifices change until the pressure drop values across the fixed orifices even out.

The model of the flow divider uses the Double-Acting Servo Cylinder, Fixed Orifice, and Orifice with Variable Area Round Holes blocks. The following figure shows the schematic diagram of the model. The Double-Acting Servo Cylinder detects the pressure drop and shifts the variable orifice control member accordingly.



Note You cannot use this block as a flow combiner. Use the Flow Divider-Combiner block instead.

Assumptions and Limitations

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops in the Double-Acting Servo Cylinder are assumed to be fully inelastic.

Parameters

- “Fixed Orifices Tab” on page 1-645
- “Variable Orifices Tab” on page 1-646

• “Servo Cylinder Tab” on page 1-648

Fixed Orifices Tab

Fixed orifice A area

The cross-sectional passage area of the fixed orifice in the P–A path. The default value is `1e-4` m^2.

Fixed orifice B area

The cross-sectional passage area of the fixed orifice in the P–B path. The default value is `1e-4` m^2.

Fixed orifice A flow discharge coefficient

Semi-empirical parameter defining the capacity of the fixed orifice in the P–A path. The value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Fixed orifice B flow discharge coefficient

Semi-empirical parameter defining the capacity of the fixed orifice in the P–B path. The value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Fixed orifice laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Fixed orifice A laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes for the fixed orifice in the P–A path. The default value is `0.999`. This parameter is visible only if the **Fixed orifice laminar transition specification** parameter is set to `Pressure ratio`.

Fixed orifice B laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes for the fixed orifice in the P–B path. The default value is `0.999`. This parameter is

visible only if the **Fixed orifice laminar transition specification** parameter is set to `Pressure ratio`.

Fixed orifice A critical Reynolds number

The maximum Reynolds number for laminar flow for the fixed orifice in the P–A path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is `10`. This parameter is visible only if the **Fixed orifice laminar transition specification** parameter is set to `Reynolds number`.

Fixed orifice B critical Reynolds number

The maximum Reynolds number for laminar flow for the fixed orifice in the P–B path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is `10`. This parameter is visible only if the **Fixed orifice laminar transition specification** parameter is set to `Reynolds number`.

Variable Orifices Tab

Diameter of round holes

Diameter of the round holes in the two identical Variable Orifice with Round Holes blocks. The default value is `0.005` m.

Number of round holes

Number of holes in each of the Variable Orifice with Round Holes blocks. The default value is 4.

Variable orifices flow discharge coefficient

Semi-empirical parameter defining the orifice capacity of the Variable Orifice with Round Holes blocks. The value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.65`.

Variable orifice A initial opening

Initial opening of the variable orifice in the P–A path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or 0 for zero-lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is `0.0025` m.

Variable orifice B initial opening

Initial opening of the variable orifice in the P–B path. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or 0 for zero-lap configuration.

The value of initial opening does not depend on the orifice orientation. The default value is `0.0025` m.

Variable orifice laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Variable orifice A laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes for the fixed orifice in the P–A path. The default value is `0.999`. This parameter is visible only if the **Variable orifice laminar transition specification** parameter is set to `Pressure ratio`.

Variable orifice B laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes for the fixed orifice in the P–B path. The default value is `0.999`. This parameter is visible only if the **Variable orifice laminar transition specification** parameter is set to `Pressure ratio`.

Variable orifice A critical Reynolds number

The maximum Reynolds number for laminar flow through the variable orifice in the P–A path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is `10`. This parameter is visible only if the **Variable orifice laminar transition specification** parameter is set to `Reynolds number`.

Variable orifice B critical Reynolds number

The maximum Reynolds number for laminar flow through the variable orifice in the P–B path. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is `10`. This parameter is visible only if the **Variable orifice laminar transition specification** parameter is set to `Reynolds number`.

Variable orifice leakage area

The total area of possible leaks in each variable orifice when it is completely closed. The main purpose of the parameter is to maintain numerical integrity of the circuit by

preventing a portion of the system from becoming isolated after the orifice is completely closed. The parameter value must be greater than 0. The default value is `1e-9` m^2.

Servo Cylinder Tab

Servo cylinder piston area

The face area of the piston in the servo cylinder. The default value is `1.6e-4` m^2.

Servo cylinder piston stroke

The full piston stroke in the servo cylinder, from one hard stop to another. The piston is located initially in the middle of the stroke and can travel half a stroke in the positive and negative direction. The default value is `0.005` m.

Servo cylinder spring rate

The spring rate of the centering springs in the servo cylinder. The default value is `1000` N/m.

Servo cylinder damping coefficient

The damping coefficient in the contact between the piston and the case of the servo cylinder. The default value is `150` N/(m/s).

Servo cylinder stop penetration coefficient

The penetration property of the piston hard stop in the servo cylinder. The hard stop is represented as absolutely inelastic, and its property is characterized by the penetration coefficient. The default value of the coefficient is `1e12` N/m/(m/s).

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

P

Hydraulic conserving port associated with the inlet port P.

A

Hydraulic conserving port associated with the outlet port A.

B

Hydraulic conserving port associated with the outlet port B.

See Also

Double-Acting Servo Cylinder | Fixed Orifice | Flow Divider-Combiner | Orifice with Variable Area Round Holes

Introduced in R2013a

Flow Divider-Combiner

Hydraulic two-path flow divider-combiner

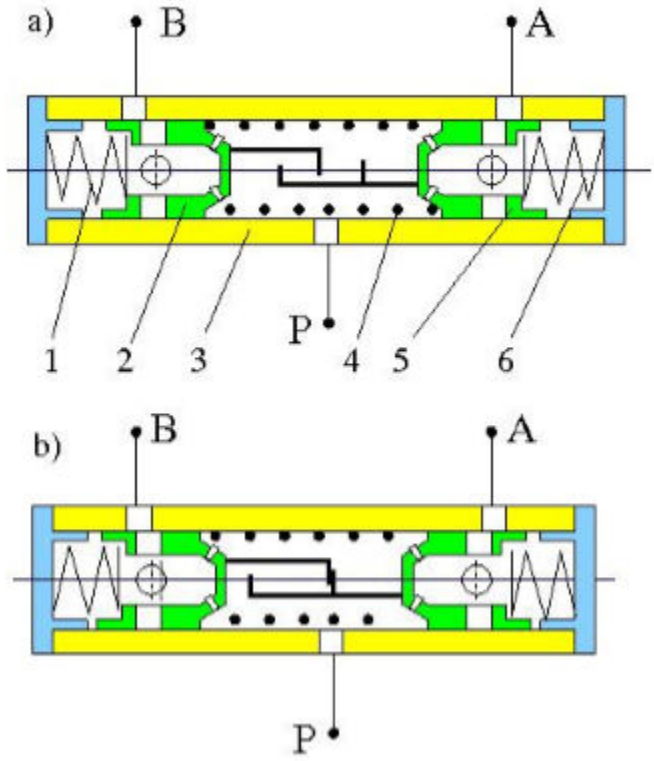
Library

Flow Control Valves

Description

The Flow Divider-Combiner block models a hydraulic valve that divides incoming flow through port P (direct flow) between two outlets, and also maintains a specified proportion between return flows through ports A and B in the total flow rate through port P. In other words, the valve works in two distinctive modes: flow divider for direct flow and flow combiner for reverse flow.

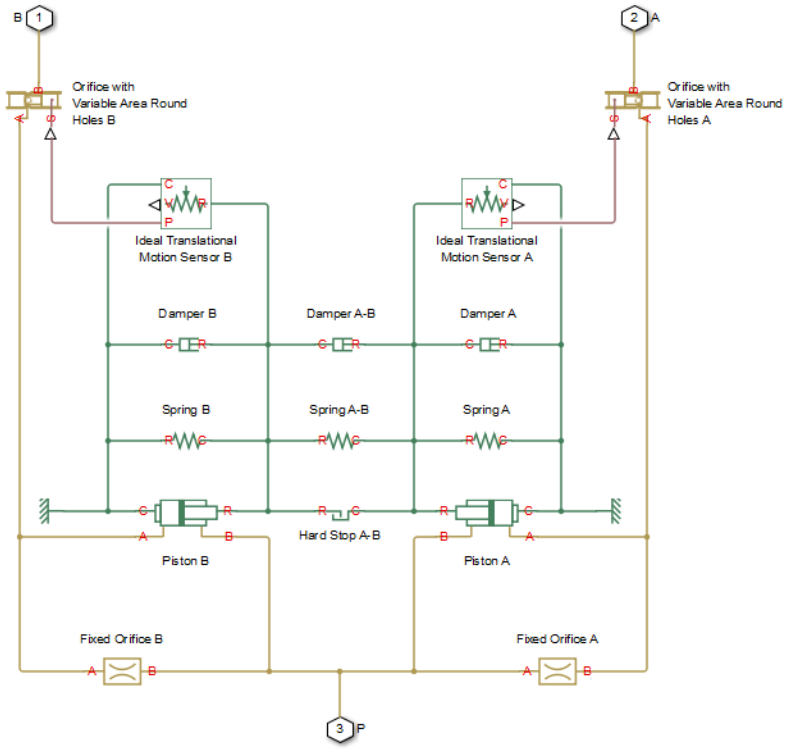
The figure shows a schematic for the flow divider-combiner valve: a) in the divider mode, and b) in the combiner mode.



The valve works as a flow divider when fluid is pumped through port P to ports A and B (schematic figure a). In this mode, fluid passes through fixed orifices in pistons 2 and 5 and through variable orifices formed by round holes in the pistons and case. The pressure differential across pistons moves them apart from each other proportionally to the piston areas and the spring 1 and 6 forces. The spring-suspended pistons and the respective variable orifices work as pressure reducing valves maintaining constant pressure drop across fixed orifices and thus keeping flow rates through them practically constant. The flow divider-combiner valve is essentially a combination of two pressure-compensated flow control valves working in parallel.

For reverse flows (schematic figure b), the pressure differential across pistons forces them against each other until the gap in the hard stop is cleared. The pistons settle at a position where pressure drops across fixed orifices are equal, thus maintaining equal flow rates through branches.

The model of the flow divider-combiner uses the Fixed Orifice, Orifice with Variable Area Round Holes, Double-Acting Hydraulic Cylinder (Simple), Translational Hard Stop, Translational Spring, and Translational Damper blocks, as shown in the block diagram.



The table explains the purpose of each model component.

Name in the block diagram	Purpose (numbers refer to the valve schematic)	Name in the actual component file
Fixed Orifice A	Fixed orifice in piston 5	fixed_orifice_A
Fixed Orifice B	Fixed orifice in piston 2	fixed_orifice_B
Piston A	Piston 5	piston_A
Piston B	Piston 2	piston_B

Name in the block diagram	Purpose (numbers refer to the valve schematic)	Name in the actual component file
Hard Stop A-B	Hard stop between pistons 2 and 5	hard_stop_A_B
Spring A	Spring 6	spring_A
Spring A-B	Spring 4	spring_A_B
Spring B	Spring 1	spring_B
Damper A	Spring 6 damping	damper_A
Damper A-B	Spring 4 damping	damper_A_B
Damper B	Spring 1 damping	damper_B
Orifice with Variable Area Round Holes A	Variable orifice created by round holes in piston 5 and the case	variable_orifice_A
Orifice with Variable Area Round Holes B	Variable orifice created by round holes in piston 2 and the case	variable_orifice_B
Ideal Translational Motion Sensor A	Measures piston 5 displacement and exports the measurement to the Orifice with Variable Area Round Holes A	sensor_A
Ideal Translational Motion Sensor B	Measures piston 2 displacement and exports the measurement to the Orifice with Variable Area Round Holes B	sensor_B

The block orientations in the model are explained by the structure section of the underlying component file, reproduced below:

```
connections
    connect(P, fixed_orifice_A.A, fixed_orifice_B.A, piston_A.B, piston_B.B);
    connect(fixed_orifice_A.B, piston_A.A, variable_orifice_A.A);
    connect(fixed_orifice_B.B, piston_B.A, variable_orifice_B.A);
    connect(B, variable_orifice_B.B);
    connect(A, variable_orifice_A.B);
    connect(reference.V, piston_A.C, spring_A.C, damper_A.C, sensor_A.C, ...
```

```
        piston_B.C, spring_B.C, damper_B.C, sensor_B.C);
    connect(piston_A.R, spring_A.R, hard_stop_A_B.C, spring_A_B.C, ...
        damper_A.R, damper_A_B.R, sensor_A.R);
    connect(piston_B.R, spring_B.R, hard_stop_A_B.R, spring_A_B.R, ...
        damper_B.R, damper_A_B.C, sensor_B.R);
    connect(sensor_A.P, variable_orifice_A.S);
    connect(sensor_B.P, variable_orifice_B.S);
end
```

Assumptions and Limitations

The block does not account for inertia, friction, and hydraulic forces. For additional assumptions and limitations, see the reference pages of the underlying member blocks.

Parameters

- “Fixed Orifices Tab” on page 1-654
- “Pistons Tab” on page 1-655
- “Springs/Dampers Tab” on page 1-656
- “Variable Orifices Tab” on page 1-657
- “Hard Stop Between Pistons Tab” on page 1-658

Fixed Orifices Tab

Fixed orifice A area

The cross-sectional passage area of the fixed orifice in piston 5 (the P–A path). The default value is `1.5e-5` m^2.

Fixed orifice B area

The cross-sectional passage area of the fixed orifice in piston 2 (the P–B path). The default value is `1.5e-5` m^2.

Fixed orifice flow discharge coefficient

Semi-empirical coefficient for fixed orifice capacity characterization. The value depends on the orifice geometrical properties, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Fixed orifice laminar transition specification

Select how the block transitions between the laminar and turbulent regimes for the fixed orifices:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Fixed orifice laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Fixed orifice critical Reynolds number** parameter.

Fixed orifice laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Fixed orifice laminar transition specification** parameter is set to `Pressure ratio`.

Fixed orifice critical Reynolds number

The maximum Reynolds number for laminar flow in the fixed orifices. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is `10`. This parameter is visible only if the **Fixed orifice laminar transition specification** parameter is set to `Reynolds number`.

Pistons Tab

Piston A area

The face area of Piston A (piston 5). The default value is `2e-4` m^2.

Piston A stroke

The full stroke of Piston A. The default value is `5` mm.

Piston A initial extension

The initial extension of Piston A. The default value is `0` m.

Piston B area

The face area of Piston B (piston 2). The default value is `2e-4` m^2.

Piston B stroke

The full stroke of Piston B. The default value is `5` mm.

Piston B initial extension

The initial extension of Piston B. The default value is `0` m.

Piston stop penetration coefficient

The penetration property of colliding bodies in the underlying cylinder blocks, which is assumed to be absolutely plastic. The default value is `1e12` s*N/m^2.

Springs/Dampers Tab

Spring A rate

Spring rate of Spring A (spring 6). The default value is `1e3` N/m.

Spring A preload

This parameter sets the initial high-priority target value for the **Deformation** variable in the underlying Spring A block. For more information, see “Variable Priority for Model Initialization” (Simscape). The default value is `0.1` m.

Damping coefficient A

Damping coefficient of Damper A (spring 6 damping). The default value is `150` N/(m/s).

Spring B rate

Spring rate of Spring B (spring 1). The default value is `1e3` N/m.

Spring B preload

This parameter sets the initial high-priority target value for the **Deformation** variable in the underlying Spring B block. For more information, see “Variable Priority for Model Initialization” (Simscape). The default value is `-0.1` m.

Damping coefficient B

Damping coefficient of Damper B (spring 1 damping). The default value is `150` N/(m/s).

Spring A-B rate

Spring rate of Spring A-B (spring 4). The default value is `1e3` N/m.

Spring A-B preload

This parameter sets the initial high-priority target value for the **Deformation** variable in the underlying Spring A-B block. For more information, see “Variable Priority for Model Initialization” (Simscape). The default value is `0.1` m.

Damping coefficient A_B

Damping coefficient of Damper A-B (spring 4 damping). The default value is `150` N/(m/s).

Variable Orifices Tab

Variable orifice A hole diameter

Diameter of the holes in the underlying Orifice with Variable Area Round Holes A block. The default value is `0.0025` m.

Variable orifice B hole diameter

Diameter of the holes in the underlying Orifice with Variable Area Round Holes B block. The default value is `0.0025` m.

Number of hole pairs in the variable orifice

Number of holes in each of the Orifice with Variable Area Round Holes blocks. The default value is `4`.

Variable orifice flow discharge coefficient

Semi-empirical parameter defining the orifice capacity of the Orifice with Variable Area Round Holes blocks. The value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Variable orifice A initial center distance

Initial opening in the underlying Orifice with Variable Area Round Holes A block. The parameter value can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is `0.0025` m, which corresponds to the position of piston 5 in the valve schematic drawing.

Variable orifice B initial center distance

Initial opening in the underlying Orifice with Variable Area Round Holes B block. The parameter value can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is `-0.0025` m, which corresponds to the position of piston 2 in the valve schematic drawing.

Variable orifice laminar transition specification

Select how the block transitions between the laminar and turbulent regimes for the variable orifices:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Variable orifice laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Variable orifice critical Reynolds number** parameter.

Variable orifice laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Variable orifice laminar transition specification** parameter is set to `Pressure ratio`.

Variable orifice critical Reynolds number

The maximum Reynolds number for laminar flow through the variable orifices. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The default value is `10`. This parameter is visible only if the **Variable orifice laminar transition specification** parameter is set to `Reynolds number`

Variable orifice leakage area

The total area of possible leaks in each variable orifice when it is completely closed. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from becoming isolated after the orifice is completely closed. The parameter value must be greater than 0. The default value is `1e-9` m^2.

Hard Stop Between Pistons Tab

Hard stop upper bound

Gap between the slider and the upper bound in the underlying Hard Stop block. The default value is `5.1` mm.

Hard stop lower bound

Gap between the slider and the lower bound in the underlying Hard Stop block. The default value is `1` mm.

Hard stop stiffness

The elastic property of colliding bodies in the hard stop. The default value is `1e8` N/m.

Hard stop damping coefficient

The dissipating property of colliding bodies in the hard stop. The default value is `150` N/(m/s).

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

P

Hydraulic conserving port associated with the inlet port P.

A

Hydraulic conserving port associated with the outlet port A.

B

Hydraulic conserving port associated with the outlet port B.

See Also

Double-Acting Hydraulic Cylinder (Simple) | Fixed Orifice | Flow Divider | Orifice with Variable Area Round Holes | Translational Damper | Translational Hard Stop | Translational Spring

Introduced in R2014b

Gas-Charged Accumulator

Hydraulic accumulator with gas as compressible medium

Library

Accumulators

Description

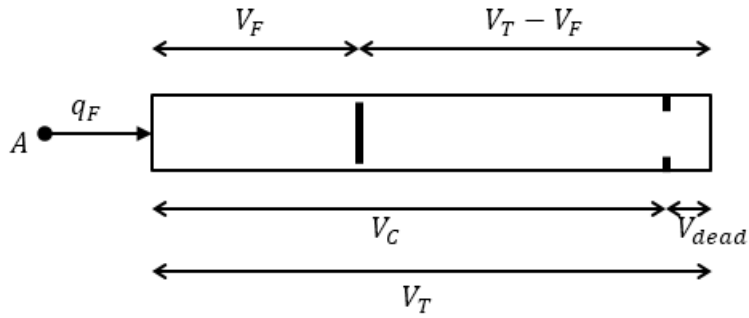
This block models a gas-charged accumulator. The accumulator consists of a precharged gas chamber and a fluid chamber. The fluid chamber is connected to a hydraulic system. The chambers are separated by a bladder, a piston, or any kind of a diaphragm.

As the fluid pressure at the accumulator inlet becomes greater than the precharge pressure, fluid enters the accumulator and compresses the gas, storing hydraulic energy. A decrease in the fluid pressure causes the gas to decompress and discharge the stored fluid into the system.

During typical operations, the pressure in the gas chamber is equal to the pressure in the fluid chamber. However, if the pressure at the accumulator inlet drops below the precharge pressure, the gas chamber becomes isolated from the system. In this situation, the fluid chamber is empty and the pressure in the gas chamber remains constant and equal to the precharge pressure. The pressure at the accumulator inlet depends on the hydraulic system to which the accumulator is connected. If the pressure at the accumulator inlet builds up to the precharge pressure or higher, fluid enters the accumulator again.

The motion of the separator between the fluid chamber and the gas chamber is restricted by two hard stops that limit the expansion and contraction of the fluid volume. The fluid volume is limited when the fluid chamber is at capacity and when the fluid chamber is empty. The hard stops are modeled with finite stiffness and damping. This means that it is

possible for the fluid volume to become negative or greater than the fluid chamber capacity, depending on the values of the hard-stop stiffness coefficient and the accumulator inlet pressure.



The diagram represents a gas-charged accumulator. The total accumulator volume (V_T) is divided into the fluid chamber on the left and the gas chamber on the right by the vertical separator. The distance between the left side and the separator defines the fluid volume (V_F). The distance between the right side and the separator defines the gas volume (V_T – V_F). The fluid chamber capacity (V_C) is less than the total accumulator volume (V_T) so that the gas volume never becomes zero.

The hard stop contact pressure is modeled with a stiffness term and a damping term. The relationship of the gas pressure and gas volume between the current state and the precharge state is given by the polytropic relation, with pressure balanced at the separator:

$$(p_G + p_A)(V_T - V_F)^k = (p_{pr} + p_A)V_T^k$$

$$p_F = p_G + p_{HS}$$

$$V_C = V_T - V_{dead}$$

$$p_{HS} = \begin{cases} K_S(V_F - V_C) + K_d q_F^+(V_F - V_C) & \text{if } V_F \geq V_C \\ K_S V_F - K_d q_F^- V_F & \text{if } V_F \leq 0 \\ 0 & \text{otherwise} \end{cases}$$

$$q_F^+ = \begin{cases} q_F & \text{if } q_F \geq 0 \\ 0 & \text{otherwise} \end{cases}$$

$$q_F^- = \begin{cases} q_F & \text{if } q_F \leq 0 \\ 0 & \text{otherwise} \end{cases}$$

where

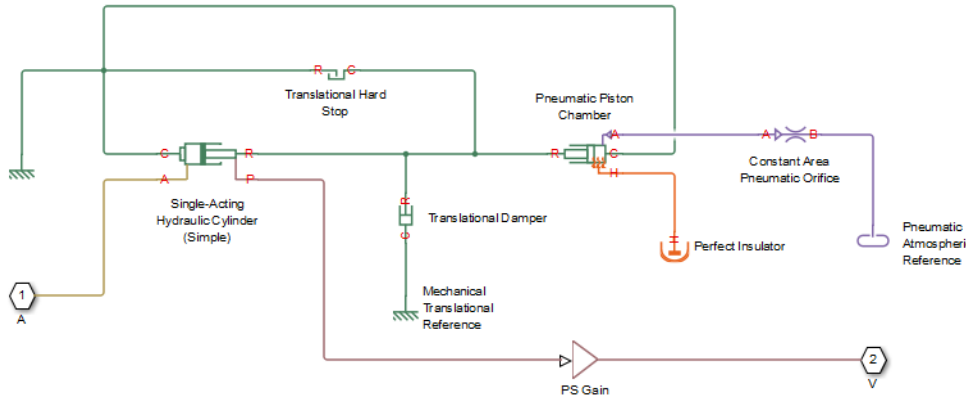
V_T	Total volume of the accumulator, including the fluid chamber and the gas chamber
V_F	Volume of fluid in the accumulator
V_{init}	Initial volume of fluid in the accumulator
V_C	Fluid chamber capacity, the difference between total accumulator volume and the gas chamber dead volume
V_{dead}	Gas chamber dead volume, a small portion of the gas chamber that remains filled with gas when the fluid chamber is at capacity
p_F	Fluid pressure (gauge) in the fluid chamber, which is equal to the pressure at the accumulator inlet
p_{pr}	Pressure (gauge) in the gas chamber when the fluid chamber is empty
p_A	Atmospheric pressure
p_G	Gas pressure (gauge) in the gas chamber
p_{HS}	Hard-stop contact pressure
K_s	Hard-stop stiffness coefficient
K_d	Hard-stop damping coefficient
k	Specific heat ratio (adiabatic index)
q_F	Fluid flow rate into the accumulator, which is positive if fluid flows into the accumulator

The flow rate into the accumulator is the rate of change of the fluid volume:

$$q_F = \frac{dV_F}{dt}$$

At $t = 0$, the initial condition is $V_F = V_{init}$, where V_{init} is the value you assign to the **Initial fluid volume** parameter.

The Gas-Charged Accumulator block does not consider loading on the separator. To model additional effects, such as the separator inertia and friction, you can construct a gas-charged accumulator as a subsystem or a composite component, similar to the block diagram below.



Basic Assumptions and Limitations

- The process in the gas chamber is assumed to be polytropic.
- Loading on the separator, such as inertia or friction, is not considered.
- Inlet hydraulic resistance is not considered.
- Fluid compressibility is not considered.

Parameters

- “Parameters Tab” on page 1-664
- “Variables Tab” on page 1-664

Parameters Tab

Total accumulator volume

Total volume of the accumulator including the fluid chamber and the gas chamber. It is the sum of the fluid chamber capacity and the minimum gas volume. The default value is `8e-3` m^3.

Minimum gas volume

Gas chamber dead volume, a small portion of the gas chamber that remains filled with gas when the fluid chamber is at capacity. A nonzero volume is necessary so that the gas pressure does not become infinite when the fluid chamber is at capacity. The default value is `4e-5` m^3.

Precharge pressure (gauge)

Pressure (gauge) in the gas chamber when the fluid chamber is empty. The default value is `10e5` Pa.

Specific heat ratio

Specific heat ratio (adiabatic index). To account for heat exchange, you can set it to a value normally between 1 and 2, depending on the properties of the gas in the gas chamber. For dry air at 20°C, this value is 1 for an isothermal process and 1.4 for an adiabatic (and isentropic) process. The default value is `1.4`.

Hard-stop stiffness coefficient

Proportionality constant of the hard-stop contact pressure with respect to the fluid volume penetrated into the hard stop. The hard stops are used to restrict the fluid volume between zero and fluid chamber capacity. The default value is `1e10` Pa/m^3.

Hard-stop damping coefficient

Proportionality constant of the hard-stop contact pressure with respect to the flow rate and the fluid volume penetrated into the hard stop. The hard stops are used to restrict the fluid volume between zero and fluid chamber capacity. The default value is `1e10` Pa*s/m^6.

Variables Tab

Accumulator flow rate

Volumetric flow rate through the accumulator port at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may

be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Volume of fluid

Volume of fluid in the accumulator at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Pressure of liquid volume

Gauge pressure in the accumulator at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Global Parameters

Atmospheric pressure

Absolute pressure of the environment. The default value is `101325` Pa.

Ports

The block has one hydraulic conserving port associated with the accumulator inlet.

The flow rate is positive if fluid flows into the accumulator.

See Also

Spring-Loaded Accumulator

Introduced in R2006a

Gas-Charged Accumulator (TL)

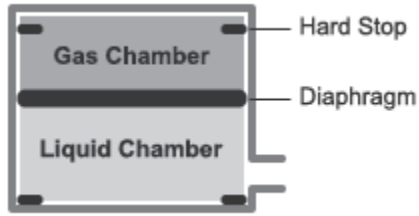
Pressurized thermal liquid container with compressed gas charge

Library

Thermal Liquid/Tanks & Accumulators

Description

The Gas-Charged Accumulator (TL) block models a pressurized thermal liquid container with a compressed gas charge. The accumulator consists of thermal liquid and gas chambers separated by a hermetic and insulated diaphragm.



Accumulator Schematic

If the inlet pressure is greater than the gas charge pressure, the liquid chamber volume increases, compressing the gas chamber. If the inlet pressure is lower than the gas charge pressure, the liquid chamber volume decreases, decompressing the gas chamber.

Hard stops limit the diaphragm motion when the liquid chamber is at capacity and when the liquid chamber is empty. The hard-stop compliance is modeled through spring and

damper forces. If the specified spring stiffness is low, the liquid volume can momentarily fall below zero or rise above capacity.

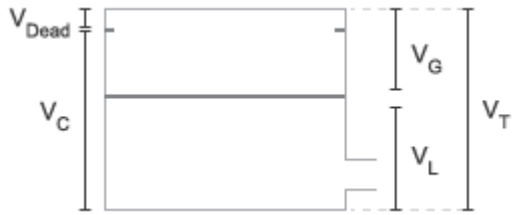
Chamber Volumes

The liquid chamber volume is the difference between the total accumulator volume and the gas chamber volume:

$$V_L = V_T - V_G,$$

where:

- V_L is the liquid chamber volume.
- V_T is the total accumulator volume.
- V_G is the gas chamber volume.



Chamber Volumes

The liquid chamber capacity is the difference between the total accumulator volume and the dead volume of gas in the accumulator at full capacity:

$$V_C = V_T - V_{Dead},$$

where:

- V_C is the liquid chamber capacity.
- V_{Dead} is the dead volume of gas at full capacity.

The gas chamber pressure and volume follow from the precharge states as described by the polytropic equation

$$p_G V_G^k = p_{pr} V_T^k,$$

where:

- p_G is the gas chamber pressure at a given time step.
- V_G is the gas chamber volume at a given time step.
- p_{pr} is the precharge gas chamber pressure when the liquid chamber is empty.
- V_T is the total liquid chamber volume.
- k is the polytropic index.

Mass Balance

The mass conservation equation in the liquid chamber is

$$V_L \rho_L \left(\frac{1}{\beta_L} \frac{dp_L}{dt} - \alpha_L \frac{dT_L}{dt} \right) + \rho_L \frac{dV_L}{dt} = \dot{m}_A,$$

where:

- ρ_L is the thermal liquid density.
- β_L is the isothermal bulk modulus.
- α_L is the isobaric thermal expansion coefficient.
- p_L is the thermal liquid pressure.
- T_L is the thermal liquid temperature.
- $\dot{m}_A$ is the thermal liquid mass flow rate into the accumulator through port A.

The time variation of the liquid chamber volume is given by the conditional equation

$$\frac{dV_L}{dt} = \begin{cases} \dfrac{\dot{p}_L}{kp_{pr}V_T^k V_G^{(-k-1)} + K_s + K_d\dot{m}^+ / \rho_L}, & V_L \geq V_C \\ \dfrac{\dot{p}_L}{kp_{pr}V_T^k V_G^{(-k-1)} + K_s - K_d\dot{m}^- / \rho_L}, & V_L \leq 0 \\ \dfrac{\dot{p}_L}{kp_{pr}V_T^k V_G^{(-k-1)}}, & \text{Else} \end{cases},$$

where:

- K_s is the hard-stop stiffness coefficient.
- K_d is the hard-stop damping coefficient.
- $\dot{m}^+$ is the mass flow rate into the liquid chamber when the accumulator diaphragm contacts the top hard stop:

  $$\dot{m}^+ = \begin{cases} \dot{m}, & \dot{m} > 0 \\ 0, & \text{Else} \end{cases},$$

- $\dot{m}^-$ is the mass flow rate from the liquid chamber when the accumulator diaphragm contacts the bottom hard stop:

  $$\dot{m}^- = \begin{cases} \dot{m}, & \dot{m} < 0 \\ 0, & \text{Else} \end{cases},$$

Momentum Balance

The momentum conservation equation in the accumulator volume is

$$p_L = p_G + p_{HS},$$

where:

- p_{HS} is the hard-stop contact pressure:

$$p_{HS} = \begin{cases} \left(V_L - V_C\right)\left(K_s + K_d \dot{m}^+ / \rho\right), & V_L \geq V_C \\ V_L \left(K_s - K_d \dot{m}^- / \rho\right), & V_L \leq 0 \\ 0, & \text{Else} \end{cases},$$

Energy Balance

The energy conservation equation in the liquid chamber volume is

$$\frac{d}{dt}\left(\rho_L u_L V_L\right) = \phi_A + \phi_H,$$

where:

- u_L is the thermal liquid specific internal energy.
- Φ_A is the energy flow rate into the liquid chamber through the accumulator inlet.
- Φ_H is the thermal energy flow rate into the liquid chamber through the accumulator wall.

Assumptions and Limitations

- Gas chamber compression is treated as a polytropic process.
- Diaphragm loading is ignored.
- Fluid inertia is ignored.

Parameters

Parameters Tab

Total accumulator volume

Combined liquid and gas volume in the accumulator. The default value is `8e-3 m^3`.

Minimum gas volume

Remnant gas volume in the accumulator in a completely filled state. The default value is `4e-5` m^3.

Precharge pressure

Initial gas charge pressure. Fluid enters the accumulator if the inlet pressure is higher than the precharge pressure. The default value is `0` MPa gauge pressure.

Specific heat ratio

Ratio of the gas specific heat at constant pressure to that at constant volume. The default value is `1.4`.

Hard-stop stiffness coefficient

Stiffness coefficient of the top and bottom accumulator hard stops. The hard stops restrict diaphragm motion between zero and the maximum liquid chamber level. The stiffness coefficient accounts for the restorative portion of the hard-stop contact forces. The default value is `1e4` MPa/m^3.

Hard-stop damping coefficient

Damping coefficients of the top and bottom accumulator hard stops. The hard stops restrict diaphragm motion between zero and the maximum liquid chamber level. The damping coefficients account for the dissipative portion of the hard-stop contact forces. The default value is `1e4` s*MPa/m^6.

Cross-sectional area at port A

Flow cross-sectional area at the accumulator inlet. The default value is `0.01` m^2.

Variables Tab

Volume of liquid

Volume of thermal liquid in the accumulator at the start of simulation. The default value is `0.005` m^3.

Mass of liquid

Mass of thermal liquid in the accumulator at the start of simulation. The default value is `5` kg.

Pressure of liquid volume

Pressure in the thermal liquid chamber at the start of simulation. The default value is `0.101325` MPa.

Temperature of liquid volume

Temperature in the thermal liquid chamber at the start of simulation. The default value is `293.15` K.

Ports

- A — Thermal liquid port representing the accumulator inlet
- H — Thermal port representing heat transfer between the liquid and the environment through the accumulator wall

See Also

Tank (TL)

Introduced in R2016a

Gate Valve

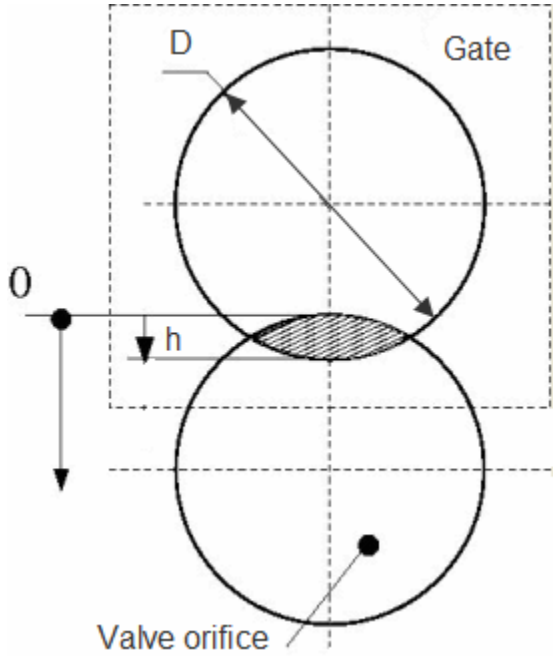
Hydraulic gate valve

Library

Flow Control Valves

Description

The Gate Valve block models a valve consisting of a round orifice in the valve housing and a flat gate, which moves perpendicular to the orifice axis. The orifice in the gate has the same diameter as the orifice in the housing. As the gate moves, it opens or closes the valve passage (shown as a shaded area in the following illustration).



The flow rate through the valve is proportional to the valve opening and to the pressure differential across the valve. The flow rate is determined according to the following equations:

$$q = C_D \cdot A(h)\sqrt{\frac{2}{\rho}}\frac{\Delta p}{\left(\Delta p^2 + p_{\mathrm{Cr}}^2\right)^{1/4}},$$

$$\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}},$$

$$h = x_0 + x$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h <= 0 \text{ or } h > 2D \\ D^2 \cdot \left(\frac{\alpha}{2} - \frac{\sin(2\alpha)}{4}\right), \alpha = a\cos\left|1 - \frac{h}{D}\right| & \text{for } 0 < h <= 2D \end{cases}$$

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Gate displacement from initial position
h	Valve opening
D	Orifice diameter
ρ	Fluid density
A_{leak}	Closed valve leakage area
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$
  $$p_{avg} = (p_A + p_B)/2$$

  where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_{H}	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

Connections A and B are hydraulic conserving ports. Connection S is a physical signal port that controls the gate displacement. The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}}$, . Positive signal at the physical signal port `S` opens the valve.

Basic Assumptions and Limitations

- No inertial effects are taken into account.

Parameters

Valve orifice diameter

The diameter of the valve orifice. The orifices in the valve housing and in the gate have the same diameter. The default value is `0.01` m.

Initial opening

The initial opening of the valve. The parameter can take both positive and negative values. The default value is `0`.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.65`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `10`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

S

Physical signal port that controls the gate displacement. The signal applied to this port is treated as translational motion, in meters.

See Also

Ball Valve | Needle Valve | Poppet Valve | Pressure-Compensated Flow Control Valve

Introduced in R2011b

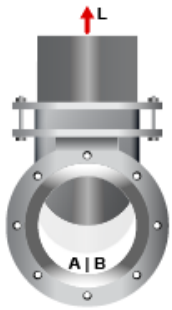
Gate Valve (G)

Valve with sliding gate as control element

Library: Simscape / Fluids / Gas / Valves & Orifices / Flow Control Valves

Description

The Gate Valve (G) block models an orifice with a translating gate (or *sluice*) as a flow control mechanism. The gate is circular and constrained by the groove of its seat to slide perpendicular to the flow. The seat is annular and its bore, part of the orifice through which the flow must pass, is sized to match the gate. The overlap of the two—the gate and the bore—determines the opening area of the valve.



Gate valve with conical seat

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Gate valves are generally quick-opening. They are most sensitive to gate displacement near the closed position, where a small displacement translates into a disproportionately

large change in opening area. Valves of this sort have too high a gain in that region to effectively throttle, or modulate, flow. They more commonly serve as binary on/off switches—often as shutoff and isolation valves—to open and close gas circuits.

Gate Mechanics

In a real valve, the gate connects by a gear mechanism to a handle. When the handle is turned from a fully closed position—by hand, say, or with the aid of an electrical actuator —the gate rises from the bore, progressively opening the valve up to a maximum. Hard stops keep the disk from breaching its minimum and maximum positions.

The block captures the motion of the disk but not the detail of its mechanics. The motion you specify as a normalized displacement at port **L**. The input (a physical signal) carries the fraction of the instantaneous displacement over its value in the fully open valve. It helps to think of displacements directly as fractions, rather than as lengths to be converted to (and from) fractions.

If the action of the handle and hard stop matter in your model, you can capture these elements separately using other Simscape blocks. A Simscape Mechanical subsystem makes a good source for the gate displacement signal. In many cases, however, it suffices to know what displacement to impart to the disk. You can usually ignore the mechanics of the valve.

Gate Position

The displacement signal allows the block to compute the instantaneous position of the gate, from which the opening of the valve follows. The opening is easily understood as a flow area but, for ease of modeling, it is often best expressed as a flow coefficient or sonic conductance. (The ``ease of modeling'' depends on the data available from the manufacturer.)

The position and displacement variables measure different things. The (instantaneous) position gives the distance of the gate to its resting place on the seat; the displacement gives only that distance to its normal (unactuated) position. The normal position, a fixed coordinate, need not be zero: the gate can be installed so that it is normally off-center with respect to the orifice. (The valve is then partially open even when it is disconnected and therefore idle.)

The normal distance between the gate and its centered position gives the *valve lift control offset*, specified in the block parameter of the same name. Think of its as the permanent

displacement given to the gate while assembling the valve. The variable displacement from port **L**, on the other hand, captures the motion of the gate during operation of the valve, after it has been assembled and installed. The instantaneous position of the gate is the sum of the two:

$$h(L) = L + h_0,$$

where:

- h is the instantaneous position of the gate, normalized against its maximum value. This variable can range from `0` to `1`, with `0` giving a maximally closed valve and `1` a fully open valve. If the calculation should return a number outside of this range, that number is set to the nearest bound (`0` if the result is negative, `1` otherwise). In other words, the normalized position *saturates* at `0` and `1`.
- L is the variable displacement of the gate, normalized against the maximum position of the same. This variable is obtained from the physical signal at port **L**. There are no restrictions on its value. You can make it smaller than `0` or greater than `1`, for example, to compensate for an equally extreme valve offset.
- h_0 is the fixed offset of the gate relative to its seat in the normal position (when the valve is disconnected and free of inputs). Its value too is normalized against the maximum position of the gate, though there is no requirement that it lie between `0` and `1`.

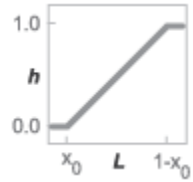
Numerical Smoothing

The normalized position, h, spans three regions. At a sufficiently small displacement, it saturates at `0` and the valve is fully closed. At a sufficiently large displacement, it saturates at `1` and the valve is fully open. In between, it varies linearly between its saturation bounds, giving a valve that is partially open.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish the simulation run, perhaps excessively so for practical use in real-time

simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized position with sharp transitions

To remove the slope discontinuities, the block smoothes them over a small portion of the opening curve. The smoothing, which adds a slight distortion at each transition, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_{\mathrm{L}} = 3\bar{h}_{\mathrm{L}}^{2} - 2\bar{h}_{\mathrm{L}}^{3}$$

and

$$\lambda_{\mathrm{R}} = 3\bar{h}_{\mathrm{R}}^{2} - 2\bar{h}_{\mathrm{R}}^{3},$$

where

$$\bar{h}_{\mathrm{L}} = \frac{h}{\Delta h^{*}}$$

and

$$\bar{h}_{\mathrm{R}} = \frac{h - \left(1 - \Delta h^{*}\right)}{\Delta h^{*}}.$$

In the equations:

- λ_{L} is the smoothing expression for the transition from the maximally closed position.

- λ_R is the smoothing expression for the transition from the fully open position.
- Δp^* is the (unitless) characteristic width of the smoothing region:

  $$\Delta h^* = \frac{1}{2} f^*,$$

  where f^* is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

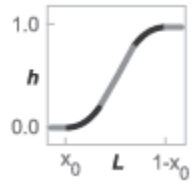
  When the smoothing factor is `0`, the normalized gate position stays in its original form —no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the gate's travel range (with the normalized gate position taking the shape of an *S*-curve).

  At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the travel range on each side (for a total smooth region of half that range).

The smoothing adds two new regions to the normalized gate position function—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$h^* = \begin{cases} 0, & h \le 0 \\ h\lambda_L, & h < \Delta h^* \\ h, & h \le 1 - \Delta h^* \\ h(1-\lambda_R) + \lambda_R, & h < 1 \\ 1 & h \ge 1 \end{cases},$$

where h^* is the smoothed valve opening area. The figure shows the effect of smoothing on the sharpness of the transitions.



Opening Area

The opening area of the valve is that of its bore adjusted for the instantaneous overlap of the gate—a function of its displacement—and leakage between its ports:

$$S = \frac{\pi D^2}{4} - S_{\mathrm{C}} + S_{\mathrm{Leak}},$$

where:

- S is the instantaneous valve opening area. This area is later smoothed to remove derivative discontinuities at the limiting valve positions.
- D is the common diameter of the gate and its bore (the two being identical). This value is obtained from the **Orifice diameter** block parameter.
- S_{C} is the area of overlap between the gate and bore, computed as a function of the gate position, h (which in turn depends on the gate displacement signal, L):

$$S_C = \frac{D^2}{2}\operatorname{acos}(h) - \frac{hD}{2}\sqrt{D^2 - h^2 D^2},$$

- S_{Leak} is the residual area that remains open after the valve has closed to its maximum. This area can be due to bore tolerances, surface defects, or an imperfect seal between the gate and its seat. This area is obtained from the **Leakage area** block parameter.

The figure shows a front view of the valve maximally closed (left), partially open (middle), and fully open (right). The parameters and variables used in the opening area calculation are shown.

Sonic Conductance

As the opening area varies during simulation, so does the mass flow rate through the valve. The relationship between the two variables, however, is indirect. The mass flow

rate is defined in terms of the valve's sonic conductance and it is this quantity that the opening area truly determines.

Sonic conductance, if you are unfamiliar with it, describes the ease with which a gas will flow when it is choked—when its velocity is at its theoretical maximum (the local speed of sound). Its measurement and calculation are covered in detail in the ISO 6358 standard (on which this block is based).

Only one value is commonly reported in valve data sheets: one taken at steady state in the fully open position. This is the same specified in the block dialog box (when the **Valve parameterization** setting is `Sonic conductance`). For values across the opening range of the valve, this maximum is scaled by the (normalized) valve opening area:

$$C(S) = \frac{S}{S_{\mathrm{Max}}} C_{\mathrm{Max}},$$

where C is sonic conductance and the subscript `Max` denotes the specified (manufacturer's) value. The sonic conductance varies linearly between C_{Max} in the fully open position and $S_{\mathrm{Leak}} \div S_{\mathrm{Max}} \times C_{\mathrm{Max}}$ in the maximally closed position—a value close to zero and due only to internal leakage between the ports.

Because sonic conductance may not be available (or the most convenient choice for your model), the block provides several equivalent parameterizations. Use the **Valve parameterization** drop-down list to select the best for the data at hand. The parameterizations are:

- `Compute from geometry`
- `Sonic conductance`
- `Cv coefficient (USCS)`
- `Kv coefficient (SI)`

The parameterizations differ only in the data that they require of you. Their mass flow rate calculations are still based on sonic conductance. If you select a parameterization other than `Sonic conductance`, then the block converts the alternate data—the (computed) opening area or a (specified) flow coefficient—into an equivalent sonic conductance.

The flow coefficients measure what is, at bottom, the same quantity—the flow rate through the valve at some agreed-upon temperature and pressure differential. They differ

only in the standard conditions used in their definition and in the physical units used in their expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Cv coefficient (USCS)`.
- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `1 bar`; it is expressed in metric units of `m^3/h`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Kv coefficient (SI)`.

If the valve parameterization is set to `Cv Coefficient (USCS)`, the sonic conductance is computed at the maximally closed and fully open valve positions from the **Cv coefficient (SI) at maximum flow** and **Cv coefficient (SI) at leakage flow** block parameters:

$$C = \left(4 \times 10^{-8} C_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where C_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`. (These are used in the mass flow rate calculations given in the Momentum Balance section.)

If the `Kv coefficient (SI)` parameterization is used instead, the sonic conductance is computed at the same valve positions (maximally closed and fully open) from the **Kv coefficient (USCS) at maximum flow** and **Kv coefficient (USCS) at leakage flow** block parameters:

$$C = \left(4.758 \times 10^{-8} K_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where K_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`.

For the `Restriction area` parameterization, the sonic conductance is computed (at the same valve positions) from the **Maximum opening area**, and **Leakage area** block parameters:

$$C = \left(0.128 \times 4 S_R / \pi\right) \mathrm{L}/(\mathrm{s\ bar}),$$

where S_R is the opening area at maximum or leakage flow. The subsonic index, m, is set to `0.5` while the critical pressure ratio, b_{cr} is computed from the expression:

$$0.41 + 0.272\left(\frac{S_R}{S_P}\right)^{0.25},$$

where the subscript `P` refers to the inlet of the connecting pipe.

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is assumed to reflect entirely in the sonic conductance of the valve (or in the data of the alternate valve parameterizations).

When the flow is choked, the mass flow rate is a function of the sonic conductance of the valve and of the thermodynamic conditions (pressure and temperature) established at the inlet. The function is linear with respect to pressure:

$$\dot{m}_{ch} = C\rho_0 p_{in}\sqrt{\frac{T_0}{T_{in}}},$$

where:

- C is the sonic conductance inside the valve. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- ρ is the gas density, here at standard conditions (subscript `0`), obtained from the **Reference density** block parameter.
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (`in`) or at standard conditions (`0`), the latter obtained from the **Reference temperature** block parameter.

When the flow is subsonic, and therefore no longer choked, the mass flow rate becomes a nonlinear function of pressure—both that at the inlet as well as the reduced value at the

outlet. In the turbulent flow regime (with the outlet pressure contained in the back-pressure ratio of the valve), the mass flow rate expression is:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{p_{\mathrm{r}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m ,$$

where:

- p_{r} is the back-pressure ratio, or that between the outlet pressure (p_{out}) and the inlet pressure (p_{in}):

  $$P_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the flow becomes choked. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

When the flow is laminar (and still subsonic), the mass flow rate expression changes to:

$$\dot{m}_{\mathrm{lam}} = C\rho_0 p_{\mathrm{in}} \left[\frac{1 - p_{\mathrm{r}}}{1 - b_{\mathrm{lam}}} \right] \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{b_{\mathrm{lam}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes (obtained from the **Laminar flow pressure ratio** block parameter). Combining the mass flow rate expressions into a single (piecewise) function, gives:

$$\dot{m} = \begin{cases} \dot{m}_{\mathrm{lam}}, & b_{\mathrm{lam}} \le p_{\mathrm{r}} < 1 \\ \dot{m}_{\mathrm{tur}}, & b_{\mathrm{cr}} \le p_{\mathrm{r}} < p_{\mathrm{lam}} \\ \dot{m}_{\mathrm{ch}}, & p_{\mathrm{r}} < b_{\mathrm{Cr}} \end{cases},$$

with the top row corresponding to subsonic and laminar flow, the middle row to subsonic and turbulent flow, and the bottom row to choked (and therefore sonic) flow.

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of gas can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**. Note that in this block the flow can reach but not exceed sonic speeds.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the gas and the wall that surrounds it. No work is done on or by the gas as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Input

L — Valve control signal, `unitless`

physical signal

Input port through which to specify the relative displacement of the gate (as a fraction of its maximum). The block uses this input to compute the position of the gate—and from it, the mass flow rate through the valve.

The gate position is the sum of this signal and the valve lift control offset (specified in the block parameter of the same name). This position should range between `0` for a maximally closed valve to `1` for a fully open valve.

The control signal range is enforced by saturating the calculated gate position at these limits. If no control signal is provided, the gate position is fixed to the specified valve lift control offset.

Conserving

`A` — Valve entrance

gas

Opening through which the working fluid can enter or exit the valve. The direction of flow depends on the pressure differential established across the valve. Both forward and backward directions are allowed.

`B` — Valve entrance

gas

Opening through which the working fluid can enter or exit the valve. The direction of flow depends on the pressure differential established across the valve. Both forward and backward directions are allowed.

Parameters

`Orifice diameter` — Diameter of the bore of the valve

`0.01 m` (default) | positive scalar in units of length

Diameter of the bore of the valve and of the gate that controls the opening area. The orifice is assumed to be constant in cross section throughout its length (from one port to the other).

`Gate displacement fraction offset` — Distance of the gate from its centered position

`0` (default) | unitless scalar

Displacement of the gate from the maximally closed position, expressed as a fraction of the travel range for the same. This displacement is the position of the gate in the normal valve position, when no control input is provided or when that input is zero. The

instantaneous gate position is calculated during simulation as the sum of this offset and the control signal specified at port **L**. The valve is partially open in its normal position when the offset is a fraction between `0` and `1`.

`Valve parameterization` — Method by which to characterize the opening of the valve

`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the **Sonic conductance** parameterization; if a different option is selected, the data specified in converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the block description for more information on the conversion.

This parameter determines which measures of valve opening you must specify—and therefore which of those measures appear as parameters in the block dialog box.

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions

`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value generally reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often

referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

Leakage area — Opening area in the maximally closed position due to sealing imperfections

`1e-12 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Opening area`.

Cross-sectional area at ports A and B — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be the same in size. The flow area specified here should ideally match those of the inlets of adjoining components.

Laminar flow pressure ratio — Pressure ratio at which the flow transitions between laminar and turbulent

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

Reference temperature — Inlet temperature used in the measurement of sonic conductance

`293.15 K` (default) | positive scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance
`1.185` (default) | positive scalar in units of density

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Smoothing factor` — Amount of smoothing to apply to the valve opening area function
`0` (default)

Amount of smoothing to apply to the opening area function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening area function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings), which tend to slow down the rate of simulation.

See Also

Ball Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

Gate Valve (TL)

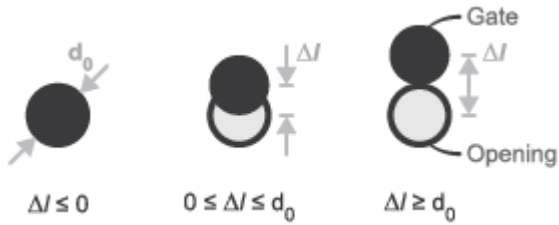
Flow control valve actuated by transverse motion of circular gate

Library

Thermal Liquid/Valves & Orifices/Flow Control Valves

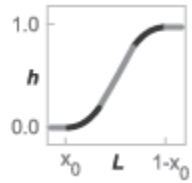
Description

The Gate Valve (TL) block represents a flow control valve with a circular opening and a circular gate. The gate moves in a direction orthogonal to the fluid flow. The opening and gate are equal in diameter. The figure shows a schematic of the gate valve in three different positions—closed, partially open, and fully open.



Gate Valve in Different Positions

A smoothing function allows the valve opening area to change smoothly between the fully closed and fully open positions. The smoothing function does this by removing the abrupt opening area changes at the zero and maximum ball positions. The figure shows the effect of smoothing on the valve opening area curve.



Opening-Area Curve Smoothing

Valve Opening Area

The block computes the valve opening area directly from valve geometry parameters using the expression

$$A = \frac{\pi d_0^2}{4} - A_{Covered},$$

where:

- A is the valve opening area.
- d_0 is the valve orifice diameter.
- $A_{Covered}$ is the portion of the valve orifice area covered by the gate:

$$A_{Covered} = \frac{d_0^2}{2}\operatorname{acos}\left(\frac{\Delta l}{d_0}\right) - \frac{\Delta l}{2}\sqrt{d_0^2 - (\Delta l)^2}$$

- Δl is the net displacement of the gate center relative to the orifice center.

$$\Delta l = \begin{cases} 0, & \left(x_0 + S_d\right) \le 0 \\ d_0, & \left(x_0 + S_d\right) \ge d_0 \\ \left(x_0 + S\right), & \text{Else} \end{cases},$$

- x_0 is the **Gate displacement offset** specified in the block dialog box.
- S_d is the gate displacement specified through physical signal input port S.

The valve opening expressions introduce undesirable discontinuities at the fully open and fully closed positions. The block eliminates these discontinuities using polynomial

expressions that smooth the transitions to and from the fully open and fully closed positions. The valve smoothing expressions are

$$\lambda_L = 3\overline{\Delta l}_L^2 - 2\overline{\Delta l}_L^3$$

and

$$\lambda_R = 3\overline{\Delta l}_R^2 - 2\overline{\Delta l}_R^3$$

where:

$$\overline{\Delta l}_L = \frac{\Delta l}{\Delta l_{smooth}}$$

and

$$\overline{\Delta l}_R = \frac{\Delta l - \left(d_0 - \Delta l_{smooth}\right)}{d_0 - \left(d_0 - \Delta l_{smooth}\right)}.$$

In the equations:

- λ_L is the smoothing expression for the fully closed portion of the valve opening curve.
- λ_R is the smoothing expression applied to the fully open portion of the valve opening curve.
- Δl_{smooth} is the gate displacement smoothing region:

  $$\Delta l_{smooth} = f_{smooth}\,\frac{d_0}{2},$$

  where f_{smooth} is a smoothing factor between 0 and 1.

The smoothed valve opening area is given by the piecewise conditional expression

$$S_R = \begin{cases} S_{Leak}, & \Delta l \le 0 \\ S_{Leak}\left(1-\lambda_L\right)+\left(A+S_{Leak}\right)\lambda_L, & \Delta l < \Delta l_{smooth} \\ A+S_{Leak}, & \Delta l \le d_0 - \Delta l_{smooth}, \\ \left(A+S_{Leak}\right)\left(1-\lambda_R\right)+\left(S_{Leak}+S_{Max}\right)\lambda_R, & \Delta l < d_0 \\ S_{Leak}+S_{Max}, & \Delta l \ge d_0 \end{cases}$$

where:

- S_R is the smoothed valve opening area.
- S_{Leak} is the valve leakage area.
- S_{Max} is the maximum valve opening area:

$$S_{Max} = \frac{\pi d_0^2}{4}.$$

Mass Balance

The mass conservation equation in the valve is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- $\dot{m}_A$ is the mass flow rate into the valve through port A.
- $\dot{m}_B$ is the mass flow rate into the valve through port B.

Energy Balance

The energy conservation equation in the valve is

$$\phi_A + \phi_B = 0,$$

where:

- ϕ_A is the energy flow rate into the valve through port A.
- ϕ_B is the energy flow rate into the valve through port B.

Momentum Balance

The momentum conservation equation in the valve is

$$p_A - p_B = \frac{\dot{m}\sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{2\rho_{Avg} C_d^2 S^2}\left[1 - \left(\frac{S_R}{S}\right)^2\right] PR_{Loss},$$

where:

- p_A and p_B are the pressures at port A and port B.
- $\dot{m}$ is the mass flow rate.
- $\dot{m}_{cr}$ is the critical mass flow rate:

  $$\dot{m}_{cr} = \mathrm{Re}_{cr}\, \mu_{Avg} \sqrt{\frac{\pi}{4} S_R}.$$

- ρ_{Avg} is the average liquid density.
- C_d is the discharge coefficient.
- S is the valve inlet area.
- PR_{Loss} is the pressure ratio:

$$PR_{Loss} = \frac{\sqrt{1 - \left(S_R/S\right)^2 \left(1 - C_d^2\right)} - C_d \left(S_R/S\right)}{\sqrt{1 - \left(S_R/S\right)^2 \left(1 - C_d^2\right)} + C_d \left(S_R/S\right)}.$$

Parameters

Parameters Tab

Orifice diameter

Diameter of the valve flow area in the fully open position. The default value is `7e-3` m.

Gate displacement offset

Gate offset from the zero position. The instantaneous gate displacement is the sum of the gate offset and input signal S. The default value is `0` m.

Leakage area

Aggregate area of all fluid leaks in the valve. The leakage area helps to prevent numerical issues due to isolated fluid network sections. For numerical robustness, set this parameter to a nonzero value. The default value is `1e-12`.

Smoothing factor

Portion of the opening-area curve to smooth expressed as a fraction. Smoothing eliminates discontinuities at the minimum and maximum flow valve positions. The smoothing factor must be between `0` and `1`. Enter a value of `0` for zero smoothing. Enter a value of `1` for full-curve smoothing. The default value is `0.01`.

Cross-sectional area at ports A and B

Flow area at the valve inlets. The inlets are assumed equal in size. The default value is `0.01` m^2.

Characteristic longitudinal length

Approximate length of the valve. This parameter provides a measure of the longitudinal scale of the valve. The default value is `0.1` m^2.

Discharge coefficient

Semi-empirical parameter commonly used as a measure of valve performance. The discharge coefficient is defined as the ratio of the actual mass flow rate through the valve to its theoretical value.

The block uses this parameter to account for the effects of valve geometry on mass flow rates. Textbooks and valve data sheets are common sources of discharge coefficient values. By definition, all values must be greater than 0 and smaller than 1. The default value is `0.7`.

Critical Reynolds number

Reynolds number corresponding to the transition between laminar and turbulent flow regimes. The flow through the valve is assumed laminar below this value and turbulent above it. The appropriate values to use depend on the specific valve geometry. The default value is `12`.

Variables Tab

Mass flow rate into port A

Mass flow rate into the component through port **A** at the start of simulation. The default value is `1 kg/s`.

Ports

- A — Thermal liquid conserving port representing valve inlet A
- B — Thermal liquid conserving port representing valve inlet B
- S — Physical signal input port for the control member displacement

See Also

Ball Valve (TL)

Introduced in R2016a

Gradual Area Change

Gradual enlargement or contraction

Library

Local Hydraulic Resistances

Description

The Gradual Area Change block represents a local hydraulic resistance, such as a gradual cross-sectional area change. The resistance represents a gradual enlargement (diffuser) if fluid flows from inlet to outlet, or a gradual contraction if fluid flows from outlet to inlet. The block is based on the Local Resistance block. It determines the pressure loss coefficient and passes its value to the underlying Local Resistance block. The block offers two methods of parameterization: by applying semi-empirical formulas (with a constant value of the pressure loss coefficient) or by table lookup for the pressure loss coefficient based on the Reynolds number.

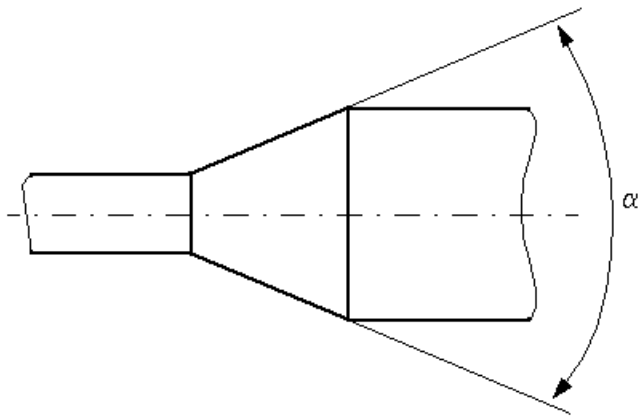
If you choose to apply the semi-empirical formulas, you provide geometric parameters of the resistance, and the pressure loss coefficient is determined according to the A.H. Gibson equations (see [1] on page 1-708 and [2] on page 1-708):

$$K_{GE} = \begin{cases} K_{cor}\left(1-\frac{A_s}{A_L}\right)^2 \cdot 2.6\sin\frac{\alpha}{2} & \text{for } 0 < \alpha <= 45^o \\ K_{cor}\left(1-\frac{A_s}{A_L}\right)^2 & \text{for } 45^o < \alpha < 180^o \end{cases}$$

$$K_{GC} = \begin{cases} K_{cor} \cdot 0.5\left(1 - \frac{A_s}{A_L}\right)^{0.75} \cdot 1.6 \sin\frac{\alpha}{2} & \text{for } 0 < \alpha <= 45^o \\ K_{cor} \cdot 0.5\left(1 - \frac{A_s}{A_L}\right)^{0.75} \cdot \sqrt{\sin\frac{\alpha}{2}} & \text{for } 45^o < \alpha < 180^o \end{cases}$$

where

K_{GE}	Pressure loss coefficient for the gradual enlargement, which takes place if fluid flows from inlet to outlet
K_{GC}	Pressure loss coefficient for the gradual contraction, which takes place if fluid flows from outlet to inlet
K_{cor}	Correction factor
A_S	Small area
A_L	Large area
α	Enclosed angle



If you choose to specify the pressure loss coefficient by a table, you have to provide a tabulated relationship between the loss coefficient and the Reynolds number. In this case, the loss coefficient is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

The pressure loss coefficient, determined by either of the two methods, is then passed to the underlying Local Resistance block, which computes the pressure loss according to the formulas explained in the reference documentation for that block. The flow regime is checked in the underlying Local Resistance block by comparing the Reynolds number to the specified critical Reynolds number value, and depending on the result, the appropriate formula for pressure loss computation is used.

The Gradual Area Change block is bidirectional and computes pressure loss for both the direct flow (gradual enlargement) and return flow (gradual contraction). If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

Connections A and B are conserving hydraulic ports associated with the block inlet and outlet, respectively.

The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- If you select parameterization by the table-specified relationship `K=f(Re)`, the flow is assumed to be turbulent.

Parameters

Model parameterization

Select one of the following methods for block parameterization:

- `By semi-empirical formulas` — Provide geometrical parameters of the resistance. This is the default method.
- `Tabulated data — loss coefficient vs. Reynolds number` — Provide tabulated relationship between the loss coefficient and the Reynolds number. The loss coefficient is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods. The table must cover both the positive and the negative flow regions.

Small diameter

Internal diameter of the small port, **A**. The default value is `0.01` m.

Large diameter

Internal diameter of the large port, **B**. The default value is `0.02` m. This parameter is used if **Model parameterization** is set to `By semi-empirical formulas`.

Cone angle

The enclosed angle. The default value is `30` deg. This parameter is used if **Model parameterization** is set to `By semi-empirical formulas`.

Correction coefficient

Correction factor used in the formula for computation of the loss coefficient. The default value is `1`. This parameter is used if **Model parameterization** is set to `By semi-empirical formulas`.

Laminar transition specification

If **Model parameterization** is set to `By semi-empirical formulas`, select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `350`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Reynolds number vector

Specify the vector of input values for Reynolds numbers as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you

must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values are `[-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]`. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Loss coefficient vector

Specify the vector of the loss coefficient values as a one-dimensional array. The loss coefficient vector must be of the same size as the Reynolds numbers vector. The default values are `[0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]`. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the resistance inlet.

B

Hydraulic conserving port associated with the resistance outlet.

References

[1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M

[2] Idelchik, I.E., *Handbook of Hydraulic Resistance*, CRC Begell House, 1994

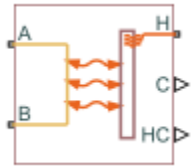
See Also

Elbow | Local Resistance | Pipe Bend | Sudden Area Change | T-junction

Introduced in R2006b

Heat Exchanger Interface (TL)

Thermal interface between a thermal liquid and its surroundings



Library

Fluid Network Interfaces/Heat Exchangers/Fundamental Components

Description

The Heat Exchanger Interface (TL) block models the pressure drop and temperature change between the thermal liquid inlet and outlet of a thermal interface. Combine with the E-NTU Heat Transfer block to model the heat transfer rate across the interface between two fluids.

Mass Balance

The form of the mass balance equation depends on the dynamic compressibility setting. If the **Fluid dynamic compressibility** parameter is set to `Off`, the mass balance equation is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- $\dot{m}_A$ and $\dot{m}_B$ are the mass flow rates into the interface through ports A and B.

If the **Fluid dynamic compressibility** parameter is set to `On`, the mass balance equation is

$$\dot{m}_A + \dot{m}_B = \left(\frac{dp}{dt} \frac{1}{\beta} - \frac{dT}{dt} \alpha \right) \rho V,$$

where:

- p is the pressure of the thermal liquid volume.
- T is the temperature of the thermal liquid volume.
- α is the isobaric thermal expansion coefficient of the thermal liquid volume.
- β is the isothermal bulk modulus of the thermal liquid volume.
- ρ is the mass density of the thermal liquid volume.
- V is the volume of thermal liquid in the heat exchanger interface.

Momentum Balance

The momentum balance in the heat exchanger interface depends on the fluid dynamic compressibility setting. If the **Fluid dynamic compressibility** parameter is set to `On`, the momentum balance factors in the internal pressure of the heat exchanger interface explicitly. The momentum balance in the half volume between port A and the internal interface node is computed as

$$p_A - p = \Delta p_{\text{Loss,A}},$$

while in the half volume between port B and the internal interface node it is computed as

$$p_B - p = \Delta p_{\text{Loss,B}},$$

where:

- p_{A} and p_{B} are the pressures at ports A and B.
- p is the pressure in the internal node of the interface volume.
- $\Delta p_{\text{Loss,A}}$ and $\Delta p_{\text{Loss,B}}$ are the pressure losses between port A and the internal interface node and between port B and the internal interface node.

If the **Fluid dynamic compressibility** parameter is set to `Off`, the momentum balance in the interface volume is computed directly between ports A and B as

$$p_A - p_B = \Delta p_{Loss,A} - \Delta p_{Loss,B}.$$

Pressure Loss Calculations

The exact form of the pressure loss terms depends on the **Pressure loss parameterization** setting in the block dialog box. If the pressure loss parameterization is set to `Constant loss coefficient`, the pressure loss in the half volume adjacent to port A is

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A \left(CP\right)_{Loss} \mathrm{Re}_L \dfrac{1}{4 D_{h,p} \rho_A S_{Min}}, & \mathrm{Re}_A \le \mathrm{Re}_L \\ \left(CP\right)_{Loss} \dfrac{\dot{m}_A \left|\dot{m}_A\right|}{4 \rho_A S_{Min}^2}, & \mathrm{Re}_A \ge \mathrm{Re}_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B \left(CP\right)_{Loss} \mathrm{Re}_L \dfrac{1}{4 D_{h,p} \rho_B S_{Min}}, & \mathrm{Re}_B \le \mathrm{Re}_L \\ \left(CP\right)_{Loss} \dfrac{\dot{m}_B \left|\dot{m}_B\right|}{4 \rho_B S_{Min}^2}, & \mathrm{Re}_B \ge \mathrm{Re}_T \end{cases},$$

where:

- μ_A and μ_B are the fluid dynamic viscosities at ports A and B.
- CP_{Loss} is the **Pressure loss coefficient** parameter specified in the block dialog box.
- Re_L is the Reynolds number upper bound for the laminar flow regime.
- Re_T is the Reynolds number lower bound for the turbulent flow regime.
- $D_{h,p}$ is the hydraulic diameter for pressure loss calculations.
- ρ_A and ρ_B are the fluid mass densities at ports A and B.
- S_{Min} is the total minimum free-flow area.

If the pressure loss parameterization is set to `Correlations for tubes`, the pressure loss in the half volume adjacent to port A is

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A \lambda \dfrac{\left(L_{press} + L_{add}\right)}{4 D_{h,p} \rho_A S_{Min}}, & \mathrm{Re}_A \le \mathrm{Re}_L \\ f_{T,A} \dfrac{\left(L_{press} + L_{add}\right)}{4 D_{h,p}} \dfrac{\dot{m}_A \left|\dot{m}_A\right|}{\rho_A S_{Min}^2}, & \mathrm{Re}_A \ge \mathrm{Re}_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B \lambda \dfrac{\left(L_{press} + L_{add}\right)}{4 D_{h,p} \rho_B S_{Min}}, & \mathrm{Re}_B \le \mathrm{Re}_L \\ f_{T,B} \dfrac{\left(L_{press} + L_{add}\right)}{4 D_{h,p}} \dfrac{\dot{m}_B \left|\dot{m}_B\right|}{\rho_B S_{Min}^2}, & \mathrm{Re}_B \ge \mathrm{Re}_T \end{cases},$$

where:

- L_{press} is the flow path length from inlet to outlet.
- L_{add} is the aggregate equivalent length of local resistances.
- $f_{\mathrm{T,A}}$ and $f_{\mathrm{T,B}}$ are the turbulent-regime Darcy friction factors at ports A and B.

The Darcy friction factor in the half volume adjacent to port A is

$$f_{T,A} = \frac{1}{\left[-1.8 \log_{10}\left(\left(\frac{6.9}{\mathrm{Re}_A}\right)^{1.11} + \frac{r}{3.7 D_{h,p}}\right)\right]^2},$$

while in the half volume adjacent to port B it is

$$f_{T,B} = \frac{1}{\left[-1.8 \log_{10}\left(\left(\frac{6.9}{\mathrm{Re}_B}\right)^{1.11} + \frac{r}{3.7 D_{h,p}}\right)\right]^2},$$

where:

- r is the internal surface absolute roughness.

If the pressure loss parameterization is set to `Tabulated data – Darcy friction factor vs. Reynolds number`, the pressure loss in the half volume adjacent to port A is

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A \lambda \dfrac{L_{press}}{4 D_{h,p}^2 \rho_A S_{Min}}, & \mathrm{Re}_A \le \mathrm{Re}_L \\ f\left(\mathrm{Re}_A\right) \dfrac{L_{press}}{4 D_{h,p}} \dfrac{\dot{m}_A \left|\dot{m}_A\right|}{\rho_A S_{Min}^2}, & \mathrm{Re}_A \ge \mathrm{Re}_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B \lambda \dfrac{L_{press}}{4 D_{h,p}^2 \rho_B S_{Min}}, & \mathrm{Re}_B \le \mathrm{Re}_L \\ f\left(\mathrm{Re}_B\right) \dfrac{L_{press}}{4 D_{h,p}} \dfrac{\dot{m}_B \left|\dot{m}_B\right|}{\rho_B S_{Min}^2}, & \mathrm{Re}_B \ge \mathrm{Re}_T \end{cases},$$

where:

- λ is the shape factor for laminar flow viscous friction.
- $f(Re_A)$ and $f(Re_B)$ are the Darcy friction factors at ports A and B. The block obtains the friction factors from tabulated data specified relative to the Reynolds number.

If the pressure loss parameterization is set to `Tabulated data – Euler number vs. Reynolds number`, the pressure loss in the half volume adjacent to port A is

$$\Delta p_{Loss,A} = \begin{cases} \dot{m}_A \mu_A \mathrm{Eu}\left(\mathrm{Re}_L\right) \mathrm{Re}_L \dfrac{1}{4 D_{h,p} \rho_A S_{Min}}, & \mathrm{Re}_A \le \mathrm{Re}_L \\ Eu\left(\mathrm{Re}_A\right) \dfrac{\dot{m}_A \left|\dot{m}_A\right|}{\rho_A S_{Min}^2}, & \mathrm{Re}_A \ge \mathrm{Re}_T \end{cases},$$

while in the half volume adjacent to port B it is

$$\Delta p_{Loss,B} = \begin{cases} \dot{m}_B \mu_B \mathrm{Eu}\left(\mathrm{Re}_L\right)\mathrm{Re}_L \dfrac{1}{4 D_{h,p} \rho_B S_{Min}}, & \mathrm{Re}_B \le \mathrm{Re}_L \\ Eu\left(\mathrm{Re}_B\right)\dfrac{\dot{m}_B \left|\dot{m}_B\right|}{\rho_B S_{Min}^2}, & \mathrm{Re}_B \ge \mathrm{Re}_T \end{cases},$$

where:

- $Eu(Re_L)$ is the Euler number at the Reynolds number upper bound for laminar flows.
- $Eu(Re_A)$ and $Eu(Re_B)$ are the Euler numbers at ports A and B. The block obtains the Euler numbers from tabulated data specified relative to the Reynolds number.

Energy Balance

The energy balance in the heat exchanger interface depends on the fluid dynamic compressibility setting. If the **Fluid dynamic compressibility** parameter is set to `On`, the energy balance is

$$\frac{dp}{dt}\frac{dU}{dp} + \frac{dT}{dt}\frac{dU}{dT} = \phi_A + \phi_B + Q_H,$$

where:

- U is the internal energy contained in the volume of the heat exchanger interface.
- Φ_A and Φ_B are the energy flow rates through ports **A** and **B** into the volume of the heat exchanger interface.
- Q_H is the heat flow rate through port **H**, representing the interface wall, into the volume of the heat exchange interface.

The internal energy derivatives are defined as

$$\frac{dU}{dp} = \left[\frac{1}{\beta}(\rho u + p) - T\alpha\right]V$$

and

$$\frac{dU}{dT} = \left[c_p - \alpha\left(u + \frac{p}{\rho}\right)\right]\rho V,$$

where u is the specific internal energy of the thermal liquid, or the internal energy contained in a unit mass of the same.

If the **Fluid dynamic compressibility** parameter is set to `Off`, the thermal liquid density is treated as a constant. The bulk modulus is then effectively infinite and the thermal expansion coefficient zero. The pressure and temperature derivatives of the compressible case vanish and the energy balance is restated as

$$\frac{dE}{dt} = \phi_A + \phi_B + Q_H,$$

where E is the total internal energy of the incompressible thermal liquid, or

$$E = \rho u V.$$

Heat Transfer Correlations

The block calculates and outputs the liquid-wall heat transfer coefficient value. The calculation depends on the **Heat transfer coefficient specification** setting in the block dialog box. If the heat transfer coefficient specification is `Constant heat transfer coefficient`, the heat transfer coefficient is simply the constant value specified in the block dialog box,

$$h_{L-W} = h_{Const},$$

where:

- $h_{\text{L-W}}$ is the liquid-wall heat transfer coefficient.
- h_{Const} is the **Liquid-wall heat transfer coefficient value** specified in the block dialog box.

For all other heat transfer coefficient parameterizations, the heat transfer coefficient is defined as the arithmetic average of the port heat transfer coefficients:

$$h_{L-W} = \frac{h_A + h_B}{2},$$

where:

- h_A and h_B are the liquid-wall heat transfer coefficients at ports A and B.

The heat transfer coefficient at port A is

$$h_A = \frac{Nu_A k_A}{D_{h,heat}},$$

while at port B it is

$$h_B = \frac{Nu_B k_B}{D_{h,heat}},$$

where:

- Nu_A and Nu_B are the Nusselt numbers at ports A and B.
- k_A and k_B are the thermal conductivities at ports A and B.
- $D_{h,heat}$ is the hydraulic diameter for heat transfer calculations.

The hydraulic diameter used in heat transfer calculations is defined as

$$D_{h,heat} = \frac{4S_{Min}L_{heat}}{S_{heat}},$$

where:

- L_{heat} is the flow path length used in heat transfer calculations.
- S_{heat} is the total heat transfer surface area.

Nusselt Number Calculations

The Nusselt number calculation depends on the **Heat transfer coefficient specification** setting in the block dialog box. If the heat transfer specification is set to `Correlations for tubes`, the Nusselt number at port A is

$$Nu_A = \begin{cases} \mathrm{Nu}_L, & \mathrm{Re}_A \le \mathrm{Re}_L \\ \dfrac{\left(f_{T,A}/8\right)\left(\mathrm{Re}_A - 1000\right)\mathrm{Pr}_A}{1 + 12.7\left(f_{T,A}/8\right)^{1/2}\left(\mathrm{Pr}_B^{2/3} - 1\right)}, & \mathrm{Re}_A \ge \mathrm{Re}_T, \end{cases}$$

while at port B it is

$$Nu_B = \begin{cases} \mathrm{Nu}_L, & \mathrm{Re}_B \le \mathrm{Re}_L \\ \dfrac{\left(f_{T,B}/8\right)\left(\mathrm{Re}_B - 1000\right)\mathrm{Pr}_B}{1 + 12.7\left(f_{T,B}/8\right)^{1/2}\left(\mathrm{Pr}_B^{2/3} - 1\right)}, & \mathrm{Re}_B \ge \mathrm{Re}_T, \end{cases}$$

where:

- Nu_L is the **Nusselt number for laminar flow heat transfer** value specified in the block dialog box.
- Pr_A and Pr_B are the Prandtl numbers at ports A and B.

If the heat transfer specification is set to `Tabulated data — Colburn data vs. Reynolds number`, the Nusselt number at port A is

$$Nu_A = j\left(\mathrm{Re}_{A,heat}\right)\mathrm{Re}_{A,heat}\,\mathrm{Pr}_A^{1/3},$$

while at port B it is

$$Nu_B = j\left(\mathrm{Re}_{B,heat}\right)\mathrm{Re}_{B,heat}\,\mathrm{Pr}_B^{1/3},$$

where:

- $j(Re_{\mathrm{A,heat}})$ and $j(Re_{\mathrm{B,heat}})$ are the Colburn numbers at ports A and B. The block obtains the Colburn numbers from tabulated data provided as a function of the Reynolds number.
- $Re_{\mathrm{A,heat}}$ and $Re_{\mathrm{B,heat}}$ are the Reynolds numbers based on the hydraulic diameters for heat transfer calculations at ports A and B. This parameter is defined at port A as

$$\mathrm{Re}_{A,heat} = \frac{\dot{m}_A D_{h,heat}}{S_{Min}\mu_A},$$

and at port B as

$$\mathrm{Re}_B = \frac{\dot{m}_B D_{h,heat}}{S_{Min}\mu_B}.$$

If the heat transfer specification is set to `Tabulated data – Nusselt number vs. Reynolds number & Prandtl number`, the Nusselt number at port A is

$$Nu_A = Nu\left(\mathrm{Re}_{A,heat}, \mathrm{Pr}_A\right),$$

while at port B it is

$$Nu_B = Nu\left(\mathrm{Re}_{B,heat}, \mathrm{Pr}_B\right).$$

Hydraulic Diameter Calculations

The hydraulic diameter used in heat transfer calculations can differ from that used in pressure loss calculations. The two parameters are different if the heated and friction perimeters are different also. For a concentric pipe heat exchanger with an annular cross-section, the hydraulic diameter for heat transfer calculations is

$$D_{h,heat} = \frac{4(\pi/4)\left(D_o^2 - D_i^2\right)}{\pi D_i} = \frac{D_o^2 - D_i^2}{D_i},$$

while the hydraulic diameter for pressure calculations is

$$D_{h,p} = \frac{4(\pi/4)\left(D_o^2 - D_i^2\right)}{\pi\left(D_i + D_o\right)} = D_o - D_i,$$

where:

- D_o is the outer annulus diameter.
- D_i is the inner annulus diameter.

Annulus Schematic

Parameters

Parameters Tab

Minimum free-flow area

Aggregate flow area free of obstacles based on the smallest tube spacing or corrugation pitch. The default value is `0.01` m^2.

Hydraulic diameter for pressure loss

Hydraulic diameter of the tubes or channels comprising the heat exchange interface. The hydraulic diameter is the ratio of the flow cross-sectional area to the channel perimeter . The default value is `0.1` m.

This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Correlations for tubes`, `Tabulated data — Darcy friction factor vs. Reynolds number`, or `Tabulated data — Euler number vs. Reynolds number`.

Laminar flow upper Reynolds number limit

Reynolds number corresponding to the upper bound of the laminar flow regime. The flow transitions to turbulent above this value. The default value is `2000`.

Turbulent flow lower Reynolds number limit

Reynolds number corresponding to the lower bound of the turbulent flow regime. The flow transitions to laminar below this value. The default value is `4000`.

Pressure loss parameterization

Parameterization used to compute the pressure loss between the inlet and outlet. You can assume a constant loss coefficient, use empirical correlations for tubes, or specify tabulated data for the Darcy friction factor or the Euler number. The default setting is `Constant loss coefficient`.

Pressure loss coefficient

Dimensionless number used to compute the pressure loss between the inlet and outlet. The pressure loss coefficient is assumed constant and the same for direct and reverse flows. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Constant loss coefficient`. The default value is `.1`.

Length of flow path from inlet to outlet

Distance traversed by the fluid from inlet to outlet. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Correlations for tubes` or `Tabulated data — Darcy friction factor vs. Reynolds number`. The default value is `1` m.

Aggregate equivalent length of local resistances

Pressure loss due to local resistances such as bends, inlets, and fittings, expressed as the equivalent length of those resistances. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Correlations for tubes`. The default value is `0.1` m.

Internal surface absolute roughness

Average height of all surface defects on the internal surface of the pipe. The surface roughness enables the calculation of the friction factor in the turbulent flow regime. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Correlations for tubes`. The default value is `15e-6` m.

Shape factor for laminar flow viscous friction

Proportionality constant between convective and conductive heat transfer in the laminar regime. The shape factor encodes the effects of component geometry on the laminar friction losses. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Correlations for tubes`. The default value is `64`.

Reynolds number vector for Darcy friction factor

M-element vector of Reynolds numbers at which to specify the Darcy friction factor. The block uses this vector to create a lookup table for the Darcy friction factor. This parameter is visible only if the **Pressure loss parameterization** parameter is set to

`Tabulated data – Darcy friction factor vs. Reynolds number`. The default vector is a 12–element vector ranging in value from `400` to `1e8`.

Darcy friction factor vector

M-element vector of Darcy friction factors corresponding to the values specified in the **Reynolds number vector for Darcy friction factor** parameter. The block uses this vector to create a lookup table for the Darcy friction factor. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Tabulated data – Darcy friction factor vs. Reynolds number`. The default vector is a 12-element vector ranging in value from `0.0214` to `0.2640`.

Reynolds number vector for Euler number

M-element vector of Reynolds numbers at which to specify the Euler number. The block uses this vector to create a lookup table for the Euler number. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Tabulated data – Euler number vs. Reynolds number`.

Euler number vector

M-element vector of Euler numbers corresponding to the values specified in the **Reynolds number vector for Euler number** parameter. The block uses this vector to create a lookup table for the Euler number. This parameter is visible only if the **Pressure loss parameterization** parameter is set to `Tabulated data – Euler number vs. Reynolds number`.

Heat transfer parameterization

Parameterization used to compute the heat transfer rate between the heat exchanger fluids. You can assume a constant loss coefficient, use empirical correlations for tubes, or specify tabulated data for the Colburn or Nusselt number. The default setting is `Constant loss coefficient`.

Heat transfer surface area

Aggregate surface area available for heat transfer between the heat exchanger fluids. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Correlation for tubes`, `Tabulated data – Colburn factor vs. Reynolds number`, or `Tabulated data – Nusselt number vs. Reynolds number & Prandtl number`. The default value is `0.4` m^2.

Liquid-wall heat transfer coefficient

Heat transfer coefficient between the thermal liquid and the heat-transfer surface. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Constant heat transfer coefficient`. The default value is `100`.

Length of flow path for heat transfer

Distance traversed by the fluid along which heat exchange takes place. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Correlation for tubes`, `Tabulated data — Colburn factor vs. Reynolds number`, or `Tabulated data — Nusselt number vs. Reynolds number & Prandtl number`. The default value is `1` m.

Nusselt number for laminar flow heat transfer

Proportionality constant between convective and conductive heat transfer in the laminar regime. This parameter enables the calculation of convective heat transfer rates in laminar flows. The appropriate value to use depends on component geometry. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Correlation for tubes`. The default value is `3.66`.

Reynolds number vector for Colburn factor

M-element vector of Reynolds numbers at which to specify the Colburn factor. The block uses this vector to create a lookup table for the Colburn number. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Tabulated data — Colburn factor vs. Reynolds number`. The default vector is `[100.0, 150.0, 1000.0]`.

Colburn factor vector

M-element vector of Colburn factors corresponding to the values specified in the **Reynolds number vector for Colburn number** parameter. The block uses this vector to create a lookup table for the Colburn factor. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Tabulated data — Colburn factor vs. Reynolds number`. The default vector is `[0.019, 0.013, 0.002]`.

Reynolds number vector for Nusselt number

M-element vector of Reynolds numbers at which to specify the Nusselt number. The block uses this vector to create a lookup table for the Nusselt number. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Tabulated data — Nusselt number vs. Reynolds number & Prandtl number`. The default vector is `[100.0, 150.0, 1000.0]`.

Prandtl number vector for Nusselt number

N-element vector of Prandtl numbers at which to specify the Nusselt number. The block uses this vector to create a lookup table for the Nusselt number. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Tabulated data — Nusselt number vs. Reynolds number & Prandtl number`. The default vector is [1.0, 10.0].

Nusselt number table, Nu(Re,Pr)

M-by-N matrix of Nusselt numbers corresponding to the values specified in the **Reynolds number vector for Nusselt number** and **Prandtl number vector for Nusselt number** parameters. The block uses this vector to create a lookup table for the Nusselt factor. This parameter is visible only when the **Heat transfer parameterization** parameter is set to `Tabulated data – Nusselt number vs. Reynolds number & Prandtl number`. The default matrix is `[3.72, 4.21; 3.75, 4.44; 4.21, 7.15]`.

Fouling factor

Empirical parameter used to quantify the increased thermal resistance due to dirt deposits on the heat transfer surface. The default value is `1e-4` m^2*K/W.

Effects and Initial Conditions

Thermal Liquid dynamic compressibility

Option to model the pressure dynamics inside the heat exchanger. Setting this parameter to `Off` removes the pressure derivative terms from the component energy and mass conservation equations. The pressure inside the heat exchanger is then reduced to the weighted average of the two port pressures.

Thermal Liquid initial temperature

Temperature of the internal volume of thermal liquid at the start of simulation.

Thermal Liquid initial pressure

Pressure of the internal volume of thermal liquid at the start of simulation.

Ports

- A — Thermal liquid conserving port representing the thermal liquid inlet
- B — Thermal liquid conserving port representing the thermal liquid outlet
- C — Physical signal output port for the thermal capacity rate of the thermal liquid
- H — Thermal conserving port associated with the thermal liquid inlet temperature
- HC — Physical signal output port for the heat transfer coefficient between the thermal liquid and the interface wall

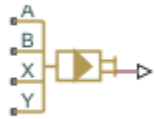
See Also

E-NTU Heat Exchanger (TL) | E-NTU Heat Exchanger (TL-TL)

Introduced in R2016a

Hydraulic 4-Port Cartridge Valve Actuator

Double-acting hydraulic valve actuator driven by four pressures

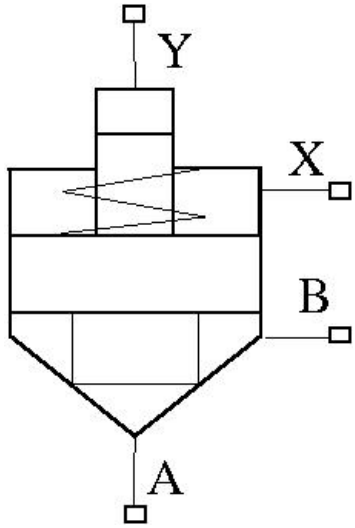


Library

Valve Actuators

Description

The Hydraulic 4-Port Cartridge Valve Actuator block represents a double-acting hydraulic valve actuator driven by four pressures. Use it as a pilot actuator for cartridge valves, pilot-operated pressure and control valves, and similar devices. A schematic diagram of the 4-port cartridge valve actuator is shown in the following illustration.



The actuator drives a valve (spool, poppet, and so on) whose position depends on pressures at ports A, B, X, and Y and the spring force. Pressures at ports A and B tend to open the valve, while pressures at control ports X and Y, together with the spring force, act to close it. The model does not account for flow consumption and loading forces, except spring and pressure forces.

Inertial properties of the actuator are accounted for by adding a first order lag between the steady-state and actual valve displacements. The lag is simulated with the combination of the PS Gain and PS Integrator blocks, enveloped by the unity feedback.

The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke.

The force acting on the poppet and the poppet displacement are computed with the following equations:

$$F_P = p_A \cdot A_A + p_B \cdot A_B - p_X \cdot A_X - p_Y \cdot A_Y$$

$$x_s = \begin{cases} 0 & \text{for } F_P <= F_{PR} \\ \left(F_P - F_{PR}\right) / k_{spr} & \text{for } F_{PR} < F_P < F_{PR} + k_{spr} \cdot x_{\max} \\ x_{\max} & \text{for } F_P >= F_{PR} + k_{spr} \cdot x_{\max} \end{cases}$$

$$x = x_s \frac{1}{Ts + 1}$$

where

F_P	Pressure force acting on the poppet
p_A, p_B, p_X, p_Y	Pressures at ports A, B, X, and Y, respectively
A_A, A_B, A_X, A_Y	Areas at ports A, B, X, and Y, respectively. Areas at ports A and Y are block parameters. Area at port X is specified by its ratio to that at port A, A_A / A_X. Area at port B is determined as $A_B = A_X - A_A$.
x	Actual poppet displacement
x_s	Steady-state poppet displacement

x_{max}	Full stroke
F_{PR}	Spring preload force
k_{spr}	Spring rate

Connections A, B, X, and Y are hydraulic conserving ports associated with the actuator ports. Connection P is a physical signal port whose output corresponds to poppet displacement. Pressures applied at ports A and B move the poppet in the positive or negative direction, depending on the value of the **Actuator orientation** parameter. Pressures at ports X and Y act in the opposite direction to pressures at ports A and B.

Basic Assumptions and Limitations

- The flow consumption associated with the valve motion is assumed to be negligible.
- The inertia, friction, and hydraulic axial forces are assumed to be small and are not taken into account.

Parameters

Port A poppet area

Effective poppet area at port A. The parameter value must be greater than zero. The default value is `3.3e-4` m^2.

Port A to port X area ratio

Ratio between poppet areas at port A and port X. The parameter value must be greater than zero. The default value is `0.66`.

Port Y piston area

Effective piston area at port Y. The parameter value must be greater than zero. The default value is `1.65e-4` m^2.

Preload force

Spring preload force. The default value is `26` N.

Spring rate

Spring rate. The default value is `1.4e4` N/m.

Poppet stroke

Maximum poppet stroke. The parameter value must be greater than zero. The default value is `5e-3` m.

Poppet-seat initial gap

Initial gap between the poppet and the seat. The parameter value must be greater than or equal to zero. The default value is `0`.

Time constant

Time constant of the first-order lag. The default value is `0.01` s.

Actuator orientation

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it moves the poppet in the positive or in the negative direction when pressure is applied at its inlet. If pressures applied at ports A and B move the poppet in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the valve inlet.

`B`

Hydraulic conserving port associated with the valve outlet.

`X`

Hydraulic conserving port associated with the valve control terminal.

Y

Hydraulic conserving port associated with the valve control terminal.

P

Physical signal port that outputs poppet displacement.

Examples

For an example of using this block, see the Injection Molding Actuation System example.

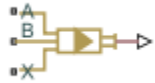
See Also

Hydraulic Cartridge Valve Actuator

Introduced in R2010a

Hydraulic Cartridge Valve Actuator

Double-acting hydraulic actuator for cartridge valves



Library

Valve Actuators

Description

Use the Hydraulic Cartridge Valve Actuator block as a pilot actuator for cartridge valves, as well as pilot-operated pressure and control valves in applications where all the forces, except spring and pressure forces, and flow consumption can be neglected. This block represents a double-acting hydraulic valve actuator driven by three pressures. The actuator drives a valve (spool, poppet, etc.) whose position depends on pressures at ports A, B, and X and the force of the spring. Pressures at ports A and B tend to open the valve, while pressure at control port X together with the spring force act to close it.

Inertial properties of the actuator are accounted for by adding a first order lag between the steady-state and actual valve displacements. The lag is simulated with the combination of the PS Gain and PS Integrator blocks, enveloped by the unity feedback.

The valve remains closed as long as the aggregate pressure force is lower than the spring preload force. The poppet is forced off its seat as the preload force is reached and moves up proportionally to pressure increase until it passes the full stroke.

Connections A, B, and X are hydraulic conserving ports associated with the actuator ports. Connection P is a physical signal port whose output corresponds to poppet displacement. Pressures applied at ports A and B move the poppet in the positive or negative direction, depending on the value of the **Actuator orientation** parameter, with pressure at port X acting in the opposite direction.

Basic Assumptions and Limitations

- The flow consumption associated with the valve motion is assumed to be negligible.
- The inertia, friction, and hydraulic axial forces are assumed to be small and are not taken into account.
- The clearances between the valve and the washers are not taken into account.

Parameters

Port A poppet area

Effective poppet area at port A. The parameter value must be greater than zero. The default value is `3.3e-4` m^2.

Port A to port X area ratio

Ratio between poppet areas at port A and port X. The parameter value must be greater than zero. The default value is `0.66`.

Preload force

Spring preload force. The default value is `26` N.

Spring rate

Spring rate. The default value is `1.4e4` N/m.

Poppet stroke

Maximum poppet stroke. The parameter value must be greater than or equal to zero. The default value is `5e-3` m.

Poppet-seat initial gap

Initial gap between the poppet and the seat. The parameter value must be greater than or equal to zero. The default value is `0`.

Time constant

Time constant of the first-order lag. The default value is `0.01` s.

Actuator orientation

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it moves the poppet in the positive or in the negative direction when pressure is applied at its inlet. If pressures applied at ports A and B move the poppet in the negative direction,

set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port associated with the valve control terminal.

P

Physical signal port that outputs poppet displacement.

See Also

2-Position Valve Actuator | 3-Position Valve Actuator | Cartridge Valve Insert | Hydraulic Double-Acting Valve Actuator | Hydraulic Single-Acting Valve Actuator | Proportional and Servo-Valve Actuator

Introduced in R2008a

Hydraulic Double-Acting Valve Actuator

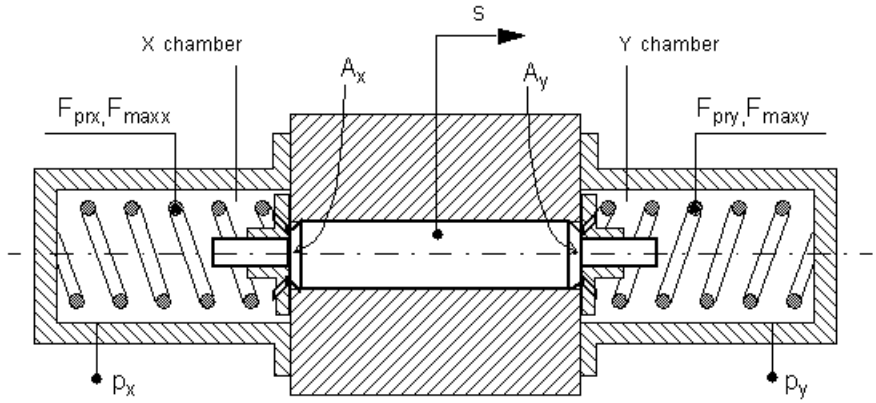
Double-acting hydraulic valve actuator

Library

Valve Actuators

Description

Use the Hydraulic Double-Acting Valve Actuator block as a pilot actuator for directional, pressure, or flow control valves in applications where all the forces, except spring force, and flow consumption can be neglected. The actuator consists of two single-acting actuators acting against each other. Each single-acting actuator consists of a piston, centering spring, and centering washer. When control pressure is applied to either hydraulic port, only one centering spring is compressed by its washer while the other butts against the valve body and exerts no force on the spool. When both control pressures are released, the springs force the washers against the valve body, and the spool centers between them. This design allows each actuator to have a different spring, preload force, and piston area.



As pressure applied to the piston develops enough force to overcome the spring preload, the piston moves to the opposite position until it reaches its maximum stroke. Pressure applied at port X shifts the valve in the *x*-direction, overcoming the spring located in the Y chamber. Pressure applied at port Y shifts the valve in the *y*-direction, overcoming the spring located in the X chamber.

The actuator is simulated according to the following equations:

$$F = p_x \bullet A_x - p_y \bullet A_y$$

$$L_x = \frac{str_x}{F_{\max x} - F_{prx}}$$

$$L_y = \frac{str_y}{F_{\max y} - F_{pry}}$$

If $F >= 0$,

$$s = \begin{cases} 0 & \text{for } F <= F_{pry} \\ L_y \bullet \left(F - F_{pry}\right) \bullet or & \text{for } F_{pry} < F < F_{\max y} \\ str_y \bullet or & \text{for } F >= F_{\max y} \end{cases}$$

If $F < 0$,

$$s = \begin{cases} 0 & \text{for } |F| <= F_{prx} \\ -L_x \cdot \left(|F| - F_{prx} \right) \cdot or & \text{for } F_{prx} < |F| < F_{\max x} \\ -str_x \cdot or & \text{for } |F| >= F_{\max x} \end{cases}$$

where

F	Force acting on the valve
s	Piston displacement
p_x	Pressure in the actuator X chamber
p_y	Pressure in the actuator Y chamber
A_x	Valve face area in the X chamber
A_y	Valve face area in the Y chamber
str_x	Valve stroke in *x*-direction
str_y	Valve stroke in *y*-direction
F_{prx}	Chamber X spring preload force
F_{maxx}	Chamber X spring maximum force
F_{pry}	Chamber Y spring preload force
F_{maxy}	Chamber Y spring maximum force
or	Actuator orientation with respect to the globally assigned positive direction. If pressure applied at port X moves the piston in positive direction, *or* equals 1. If pressure applied at port X moves the piston in negative direction, *or* equals –1.

Connections X and Y are hydraulic conserving ports associated with the valve chambers. Connection P is a physical signal port whose output corresponds to piston displacement. Pressure applied at port X moves the piston in the positive or negative direction depending on the value of the **Actuator orientation** parameter.

Basic Assumptions and Limitations

- The flow consumption associated with the valve motion is assumed to be negligible.

- The inertia, friction, and hydraulic axial forces are assumed to be small and are not taken into account.
- The clearances between the valve and the washers are not taken into account.

Parameters

Piston area at port X

Effective piston area at port X. The parameter value must be greater than zero. The default value is `2e-4` m^2.

Piston area at port Y

Effective piston area at port Y. The parameter value must be greater than zero. The default value is `2e-4` m^2.

Preload force at port X

Spring preload force at port X. The default value is `0`.

Preload force at port Y

Spring preload force at port Y. The default value is `0`.

Spring maximum force at port X

Chamber X spring maximum force. The parameter value must be greater than the spring preload force. The default value is `50` N.

Spring maximum force at port Y

Chamber Y spring maximum force. The parameter value must be greater than the spring preload force. The default value is `50` N.

Piston stroke at port X

Piston stroke in chamber X. The parameter value must be greater than or equal to zero. The default value is `5e-3` m.

Piston stroke at port Y

Piston stroke in chamber Y. The parameter value must be greater than or equal to zero. The default value is `5e-3` m.

Actuator orientation

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it moves the piston in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port X moves the piston in the negative direction, set the

parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

X

Hydraulic conserving port associated with the valve X chamber.

Y

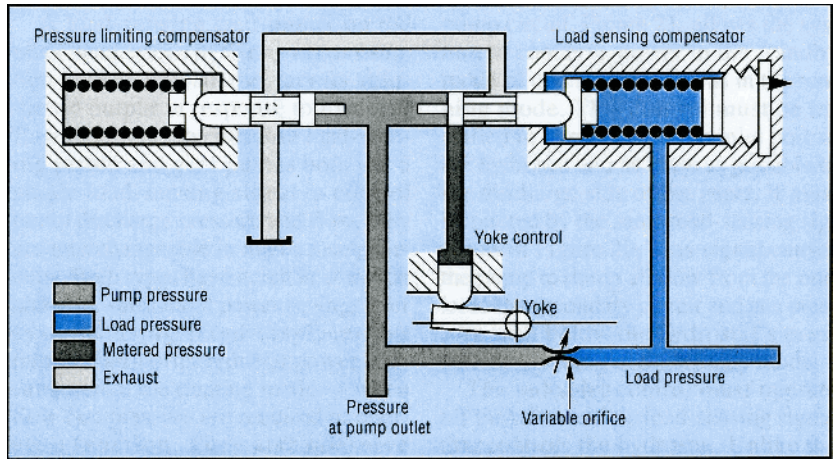
Hydraulic conserving port associated with the valve Y chamber.

P

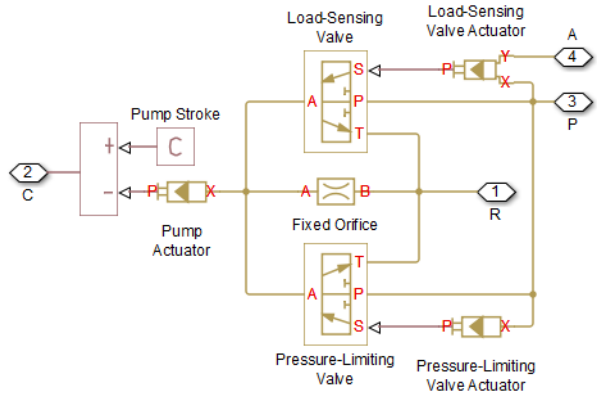
Physical signal port that outputs piston displacement.

Examples

The following illustration shows a typical control unit of a variable-displacement pump that provides load sensing and pressure limiting (see [1] on page 1-739). In the unit, the load-sensing compensator varies the pump displacement to maintain a preset pressure differential across the variable orifice, while the pressure-limiting compensator does not allow the pump pressure to exceed the pressure limit.



The Hydraulic Motor Driven by Load-Sensing Variable-Displacement Pump example implements this type of control. The next illustration shows the schematic of the Control block in the example model.



There are three hydraulic valve actuators in the model:

- Pressure-Limiting Valve Actuator — A single-acting actuator that controls the Pressure-Limiting Valve.
- Pump Actuator — A single-acting valve actuator that acts on the pump displacement control device (yoke control).

- Load-Sensing Valve Actuator — A double-acting valve actuator that controls the Load-Sensing Valve. Its output is proportional to the difference between the pump pressure (port P) and the load pressure (port A).

Open the example model to see the parameter settings for the blocks.

References

[1] F. Yeapple, *Fluid Power Design Handbook*, Marcel Dekker, Inc., 1995

See Also

2-Position Valve Actuator | 3-Position Valve Actuator | Hydraulic Single-Acting Valve Actuator | Proportional and Servo-Valve Actuator

Introduced in R2007a

Hydraulic Fluid

Working fluid properties, set by selecting from list of predefined fluids

Library

Hydraulic Utilities

Description

The Hydraulic Fluid block lets you specify the type of hydraulic fluid used in a loop of hydraulic blocks. It provides the hydraulic fluid properties, such as kinematic viscosity, density, and bulk modulus, for all the hydraulic blocks in the loop. These fluid properties are assumed to be constant during simulation time. The density is determined by the type of fluid, while kinematic viscosity additionally requires that the temperature is specified.

The bulk modulus value shown in the block dialog box is the bulk modulus of pure liquid, and is determined by the type of fluid and by the temperature. When the fluid properties are used in hydraulic blocks, such as Constant Volume Hydraulic Chamber or Variable Hydraulic Chamber, the fluid is represented as a mixture of liquid and a small amount of entrained, nondissolved gas, which is specified in the Hydraulic Fluid block as **Relative amount of trapped air**. The mixture bulk modulus in these blocks is determined as:

$$E = E_l \frac{1+\alpha\left(\frac{p_a}{p_a+p}\right)^{1/n}}{1+\alpha\frac{p_a^{1/n}}{n\cdot(p_a+p)^{\frac{n+1}{n}}}E_l}$$

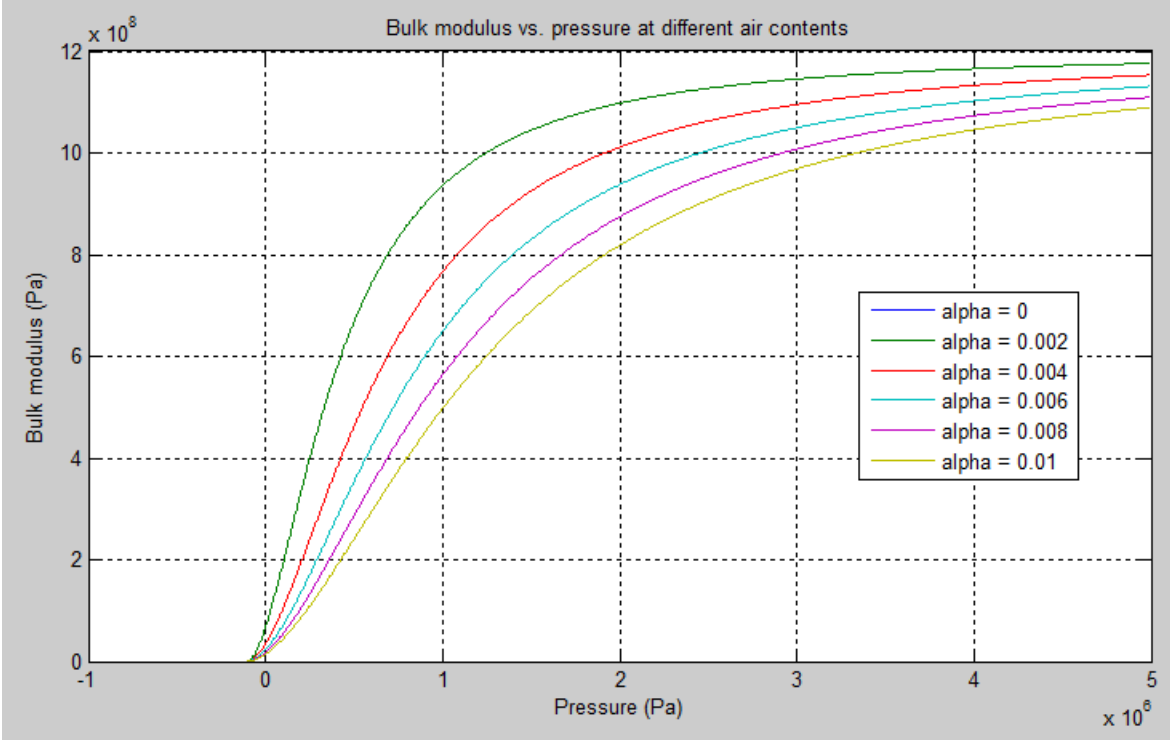
where

E_l	Pure liquid bulk modulus
p_α	Atmospheric pressure
α	Relative gas content at atmospheric pressure, α = V_G/V_L
V_G	Gas volume at atmospheric pressure
V_L	Volume of liquid
n	Gas-specific heat ratio

The main objective of representing fluid as a mixture of liquid and gas is to introduce an approximate model of cavitation, which takes place in a chamber if pressure drops below fluid vapor saturation level. As it is seen in the graph below, the bulk modulus of a

mixture decreases at $p \to p_a$, thus considerably slowing down further pressure change.

At high pressure, $p >> p_a$, a small amount of nondissolved gas has practically no effect on the system behavior.



Cavitation is an inherently thermodynamic process, requiring consideration of multiple-phase fluids, heat transfers, and so on, and as such cannot be accurately simulated with Simscape Fluids software. But the simplified version implemented in the block is good enough to signal if pressure falls below dangerous level, and to prevent computation failure that normally occurs at negative pressures.

The Hydraulic Fluid block offers a selection of predefined fluids. See “Examples” on page 1-745 for how you can get information on the fluid properties used in the block. Once you select a fluid name, you can also specify the temperature of the fluid and the relative amount of entrained, nondissolved gas.

The Hydraulic Fluid block has one port. You can connect it to a hydraulic diagram by branching a connection line off the main line and connecting it to the port. When you connect the Hydraulic Fluid block to a hydraulic line, the software automatically identifies the hydraulic blocks connected to the particular loop and propagates the hydraulic fluid properties to all the hydraulic blocks in the loop.

Each topologically distinct hydraulic loop in a diagram requires the properties of its working fluid to be specified. You can specify these properties either by selecting a predefined fluid using a Hydraulic Fluid block, or by specifying parameters of a custom fluid using a Custom Hydraulic Fluid block, which is available in the Simscape Foundation library. If no Hydraulic Fluid block or Custom Hydraulic Fluid block is attached to a loop, the hydraulic blocks in this loop use the default fluid, which is equivalent to fluid defined by a Custom Hydraulic Fluid block with the default parameter values.

Parameters

Hydraulic fluid

Hydraulic fluid type. Select one of the predefined fluids:

- `Skydrol LD-4` (default)
- `Skydrol 500B-4`
- `Skydrol-5`
- `HyJet-4A`
- `Fluid MIL-F-83282`
- `Fluid MIL-F-5606`
- `Fluid MIL-F-87257`
- `Oil-10W`
- `Oil-30W`
- `Oil-50W`
- `Oil SAE-30`
- `Oil SAE-50`
- `Transmission fluid ATF (Dexron III)`
- `ISO VG 22 (ESSO UNIVIS N 22)`
- `ISO VG 32 (ESSO UNIVIS N 32)`
- `ISO VG 46 (ESSO UNIVIS N 46)`
- `Brake fluid DOT3`
- `Brake fluid DOT4`
- `Brake fluid DOT5`

- `Gasoline`
- `Diesel fuel`
- `Jet fuel`
- `Water-Glycol 60/40`
- `Water`

Relative amount of trapped air

Amount of entrained, nondissolved gas in the fluid. The amount is specified as the ratio of gas volume at normal conditions to the fluid volume in the chamber. Therefore, the parameter value must be less than 1. In practice, the relative amount of trapped air is always greater than 0. If set to 0, ideal fluid is assumed. The default value is `0.005`.

System temperature

Fluid temperature (C). The default value is `60`.

Viscosity derating factor

Proportionality coefficient that you can use to adjust fluid viscosity, if needed. Specify a value between 0.5 and 1.5. The default value is `1`.

Pressure below absolute zero

Determines how the block handles the out-of-range assertion during simulation:

- `Error` — If the pressure falls below absolute zero, the simulation stops and you get an error message. This is the default.
- `Warning` — If the pressure falls below absolute zero, you get a warning but the simulation continues. Use this option when modeling systems where cavitation can occur in extreme cases.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Hydraulic fluid**
- **Pressure below absolute zero**

All other block parameters are available for modification.

Ports

The block has one hydraulic conserving port.

Examples

You can get information on the fluids and their properties through the MATLAB® command line. In the following example, the first command brings you the list of available fluids, and the second command plots the properties of a selected fluid from the list, in this case, Skydrol LD-4.

1. In the MATLAB Command Window, type:

   ```
   props = sh_stockfluidproperties
   ```

   The system responds with a list of available fluids:

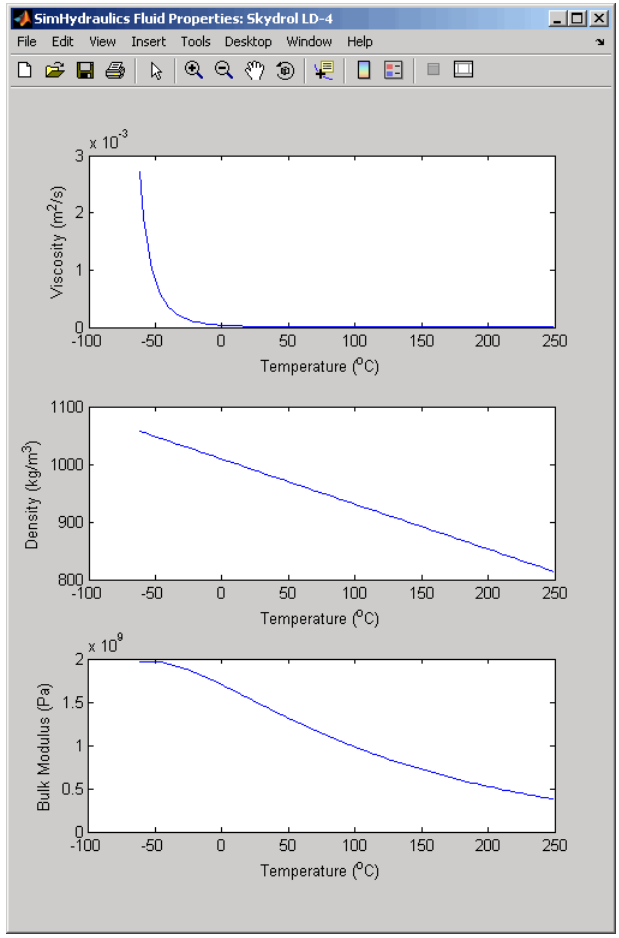
   ```
   props =

        skydrol_ld_4: [1x1 struct]
       skydrol_500_4: [1x1 struct]
           skydrol_5: [1x1 struct]
              hy_jet: [1x1 struct]
            f_83282: [1x1 struct]
             f_5606: [1x1 struct]
            f_87257: [1x1 struct]
            oil_10w: [1x1 struct]
            oil_30w: [1x1 struct]
            oil_50w: [1x1 struct]
          oil_sae_30: [1x1 struct]
          oil_sae_50: [1x1 struct]
          atf_dexron: [1x1 struct]
           iso_vg_32: [1x1 struct]
             gasoline: [1x1 struct]
         diesel_fuel: [1x1 struct]
            jet_fuel: [1x1 struct]
        water_glycol: [1x1 struct]
   ```

2. To plot the properties of the first fluid in the list, Skydrol LD-4, type:

   ```
   props.skydrol_ld_4.plot()
   ```

   The plot window opens:



Fluid properties for the Skydrol family of hydraulic fluids were obtained from literature provided by the manufacturer, Solutia, Inc. More information is available on their website at: http://www.skydrol.com.

See Also

Custom Hydraulic Fluid

Introduced in R2006a

Hydraulic Pipeline

Hydraulic pipeline with resistive and fluid compressibility properties

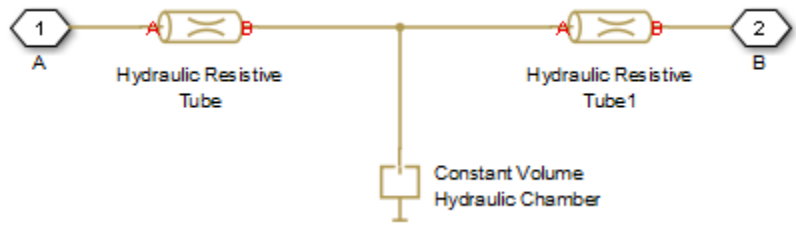
Library

Pipelines

Description

The Hydraulic Pipeline block models hydraulic pipelines with circular and noncircular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia and cannot be used for predicting effects like water hammer or changes in pressure caused by fluid acceleration.

The model is built of Simscape Foundation library building blocks and its schematic diagram is shown below.



The Hydraulic Resistive Tube blocks account for friction losses, while the Constant Volume Hydraulic Chamber block accounts for fluid compressibility. By using the block parameters, you can set the model to simulate pipeline with rigid or flexible walls, including simulation of hydraulic hoses with elastic and viscoelastic properties.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia is not taken into account.

Parameters

Pipe cross section type

The type of pipe cross section: `Circular` or `Noncircular`. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is `Circular`.

Pipe internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to `Circular`. The default value is `0.01` m.

Noncircular pipe cross-sectional area

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `1e-4` m^2.

Noncircular pipe hydraulic diameter

Hydraulic diameter of the pipe cross section. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `0.0112` m.

Geometrical shape factor

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1 on page 1-751]. The default value is `64`, which corresponds to a pipe with a circular cross section.

Pipe length

Pipe geometrical length. The default value is `5` m.

Aggregate equivalent length of local resistances

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such

as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. This length is added to the geometrical pipe length only for hydraulic resistance computation. The fluid volume depends on pipe geometrical length only. The default value is `1` m.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Pipe wall type

The parameter is available only for circular pipes and can have one of two values: `Rigid` or `Flexible`. If the parameter is set to `Rigid`, wall compliance is not taken into account, which can improve computational efficiency. The value `Flexible` is recommended for hoses and metal pipes where wall compliance can affect the system behavior. The default value is `Rigid`.

Static pressure-diameter coefficient

Coefficient that establishes relationship between the pressure and the internal diameter at steady-state conditions. This coefficient can be determined analytically for cylindrical metal pipes or experimentally for hoses. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `2e-10` m/Pa.

Viscoelastic process time constant

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `0.008` s.

Specific heat ratio

Gas-specific heat ratio for the Constant Volume Hydraulic Chamber block. The default value is `1.4`.

Initial pressure

Initial pressure in the pipe. This parameter specifies the initial condition for use in computing the block's initial state at the beginning of a simulation run. For more information, see “Initial Conditions Computation” (Simscape). The default value is `0`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Pipe cross section type**
- **Pipe wall type**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the values of the **Pipe cross section type** and **Pipe wall type** parameters at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

See Also

Hydraulic Resistive Tube | Linear Hydraulic Resistance | Segmented Pipeline

Introduced in R2006a

Hydraulic Pipe LP

Hydraulic pipeline with resistive, fluid compressibility, and elevation properties

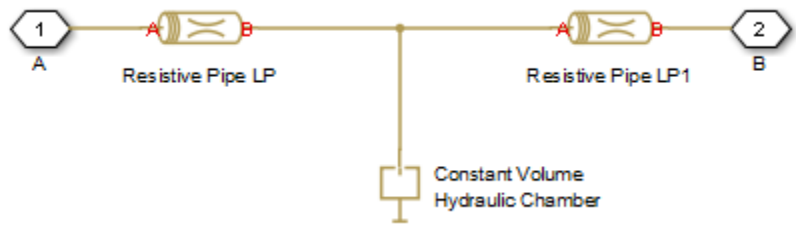
Library

Low-Pressure Blocks

Description

The Hydraulic Pipe LP block models hydraulic pipelines with circular and noncircular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia and cannot be used for predicting effects like water hammer or changes in pressure caused by fluid acceleration.

The model is a structural model and its schematic diagram is shown below.



The Resistive Pipe LP blocks account for friction losses, while the Constant Volume Hydraulic Chamber block accounts for fluid compressibility. To reduce model complexity, you can use this block to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, associated with the pipe. You must convert the resistances into their equivalent lengths, and then sum up all

the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length. By using the block parameters, you can set the model to simulate pipeline with rigid or flexible walls, including simulation of hydraulic hoses with elastic and viscoelastic properties.

The difference in elevation between ports A and B is distributed evenly between pipe segments.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia is not taken into account.

Parameters

- “Basic Parameters Tab” on page 1-753
- “Wall Compliance Tab” on page 1-755
- “Vertical Position Tab” on page 1-755

Basic Parameters Tab

Pipe cross section type

The type of pipe cross section: `Circular` or `Noncircular`. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is `Circular`.

Pipe internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to `Circular`. The default value is `0.01` m.

Noncircular pipe cross-sectional area

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `1e-4` m^2.

Noncircular pipe hydraulic diameter

Hydraulic diameter of the pipe cross section. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `0.0112` m.

Geometrical shape factor

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1 on page 1-751]. The default value is `64`, which corresponds to a pipe with a circular cross section.

Pipe length

Pipe geometrical length. The default value is `5` m.

Aggregate equivalent length of local resistances

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. This length is added to the geometrical pipe length only for hydraulic resistance computation. The fluid volume depends on pipe geometrical length only. The default value is `1` m.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Laminar flow upper Reynolds number

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower Reynolds number

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Initial pressure

Gauge pressure in the pipe at time zero. The default value is `0` Pa.

Wall Compliance Tab

Pipe wall type

The parameter is available only for circular pipes and can have one of two values: `Rigid` or `Flexible`. If the parameter is set to `Rigid`, wall compliance is not taken into account, which can improve computational efficiency. The value `Flexible` is recommended for hoses and metal pipes where wall compliance can affect the system behavior. The default value is `Rigid`.

Static pressure-diameter coefficient

Coefficient that establishes relationship between the pressure and the internal diameter at steady-state conditions. This coefficient can be determined analytically for cylindrical metal pipes or experimentally for hoses. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `2e-12` m/Pa.

Viscoelastic process time constant

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `0.01` s.

Specific heat ratio

Gas-specific heat ratio for the Constant Volume Hydraulic Chamber block. The default value is `1.4`. If **Pipe cross section type** is set to `Noncircular`, then this is the only parameter on the Wall Compliance tab.

Vertical Position Tab

Port A elevation from reference plane

Vertical position of port **A** with respect to a reference plane. The reference plane is assumed to be the same as that used in the **Port B elevation from reference plane** parameter. The default value is `0`m.

Port B elevation from reference plane

Vertical position of port **B** with respect to a reference plane. The reference plane is assumed to be the same as that used in the **Port A elevation from reference plane** parameter. The default value is `0`m.

Gravitational acceleration

Value of the gravitational acceleration constant (g). The block uses this value to compute the pressure differential between the ports due to a change in elevation. The default value is `9.80655` m/s^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Pipe cross section type**
- **Pipe wall type**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the values of the **Pipe cross section type** and **Pipe wall type** parameters at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the pipe inlet.

`B`

Hydraulic conserving port associated with the pipe outlet.

References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

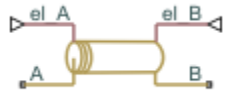
See Also

Hydraulic Resistive Tube | Hydraulic Pipe LP with Variable Elevation | Hydraulic Pipeline | Linear Hydraulic Resistance | Resistive Pipe LP with Variable Elevation | Resistive Pipe LP | Segmented Pipe LP | Segmented Pipeline

Introduced in R2009a

Hydraulic Pipe LP with Variable Elevation

Hydraulic pipeline with resistive, fluid compressibility, and variable elevation properties



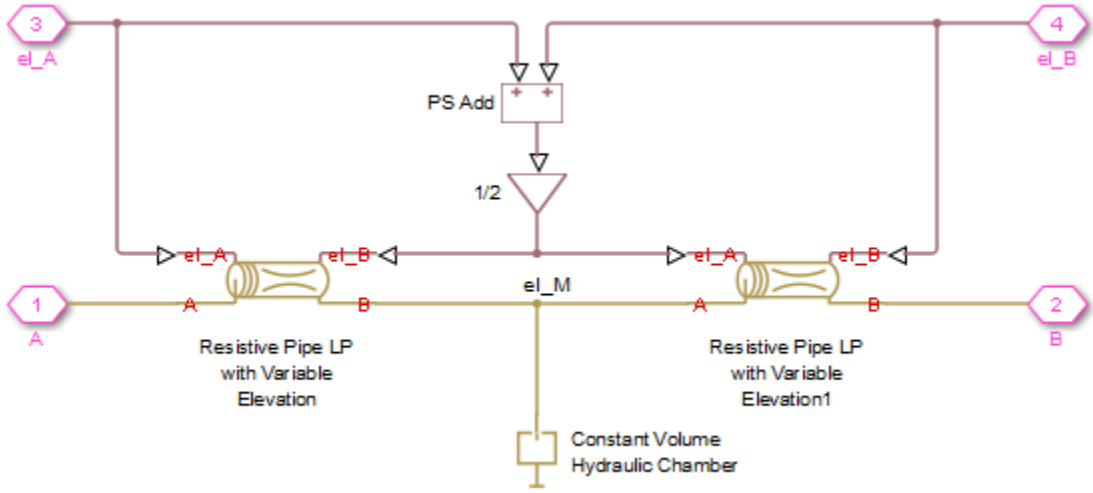
Library

Low-Pressure Blocks

Description

The Hydraulic Pipe LP with Variable Elevation block models hydraulic pipelines with circular and noncircular cross sections. The block accounts for friction loss along the pipe length and for fluid compressibility. The block does not account for fluid inertia and cannot be used for predicting effects like water hammer or changes in pressure caused by fluid acceleration. Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane. The elevations are provided through respective physical signal inputs.

The model is a structural model and its schematic diagram is shown below.



The Resistive Pipe LP with Variable Elevation blocks account for friction losses, while the Constant Volume Hydraulic Chamber block accounts for fluid compressibility. To reduce model complexity, you can use this block to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, associated with the pipe. You must convert the resistances into their equivalent lengths, and then sum up all the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length. By using the block parameters, you can set the model to simulate pipeline with rigid or flexible walls, including simulation of hydraulic hoses with elastic and viscoelastic properties.

The difference in elevation between ports A and B is assumed to be distributed evenly between pipe segments. Thus, the elevation of the pipe middle point is computed as

$$el_M = \frac{el_A + el_B}{2}$$

where

el_M	Elevation of the pipe middle point
el_A, *el_B*	Elevations of the pipe ends A and B, respectively

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia is not taken into account.
- The pipe elevation is distributed evenly along the pipe length.

Parameters

- "Basic Parameters Tab" on page 1-760
- "Wall Compliance Tab" on page 1-761

Basic Parameters Tab

Pipe cross section type

The type of pipe cross section: `Circular` or `Noncircular`. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is `Circular`.

Pipe internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to `Circular`. The default value is `0.01` m.

Noncircular pipe cross-sectional area

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `1e-4` m^2.

Noncircular pipe hydraulic diameter

Hydraulic diameter of the pipe cross section. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `0.0112` m.

Geometrical shape factor

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for

rectangle (2:1), and so on [1 on page 1-751]. The default value is `64`, which corresponds to a pipe with a circular cross section.

Pipe length

Pipe geometrical length. The default value is `5` m.

Aggregate equivalent length of local resistances

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. This length is added to the geometrical pipe length only for hydraulic resistance computation. The fluid volume depends on pipe geometrical length only. The default value is `1` m.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Laminar flow upper Reynolds number limit

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower Reynolds number limit

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Gravitational acceleration

Value of the gravitational acceleration constant (g). The block uses this parameter to calculate time changes in pressure due to time changes in elevation. The default value is `9.80655` m/s^2.

Initial pressure

Gauge pressure in the pipe at time zero. The default value is `0` Pa.

Wall Compliance Tab

Pipe wall type

The parameter is available only for circular pipes and can have one of two values: `Rigid` or `Flexible`. If the parameter is set to `Rigid`, wall compliance is not taken

into account, which can improve computational efficiency. The value `Flexible` is recommended for hoses and metal pipes where wall compliance can affect the system behavior. The default value is `Rigid`.

Static pressure-diameter coefficient

Coefficient that establishes relationship between the pressure and the internal diameter at steady-state conditions. This coefficient can be determined analytically for cylindrical metal pipes or experimentally for hoses. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `2e-12` m/Pa.

Viscoelastic process time constant

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `0.01` s.

Specific heat ratio

Gas-specific heat ratio for the Constant Volume Hydraulic Chamber block. The default value is `1.4`. If **Pipe cross section type** is set to `Noncircular`, then this is the only parameter on the Wall Compliance tab.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the pipe inlet.

`B`

Hydraulic conserving port associated with the pipe outlet.

`el_A`

Physical signal input port that controls pipe elevation at port A.

`el_B`

Physical signal input port that controls pipe elevation at port B.

Examples

For an example of using this block, see the Aircraft Fuel Supply System with Three Tanks example.

References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

See Also

Hydraulic Resistive Tube | Hydraulic Pipe LP | Hydraulic Pipeline | Linear Hydraulic Resistance | Resistive Pipe LP | Resistive Pipe LP with Variable Elevation | Segmented Pipe LP | Segmented Pipeline

Introduced in R2010a

Hydraulic Single-Acting Valve Actuator

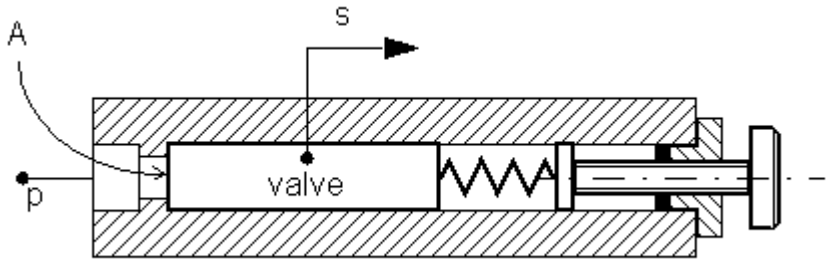
Single-acting hydraulic valve actuator

Library

Valve Actuators

Description

Use the Hydraulic Single-Acting Valve Actuator block as a pilot actuator for directional, pressure, or flow control valves in applications where all the forces, except spring force, and flow consumption can be neglected.



The actuator consists of a piston and a spring. The spring, which can be preloaded, tends to keep the piston at the initial position. As pressure applied to the piston develops enough force to overcome the spring preload, the piston moves to the opposite position until it reaches its maximum stroke.

The actuator is simulated according to the following equations:

$$F = p \bullet A$$

$$L = \frac{stroke}{F_{\max} - F_{pr}}$$

$$s = \begin{cases} 0 & \text{for } F <= F_{pr} \\ L \bullet \left(F - F_{pr} \right) \bullet or & \text{for } F_{pr} < F < F_{\max} \\ stroke \bullet or & \text{for } F >= F_{\max} \end{cases}$$

where

p	Pressure applied to the piston
s	Piston displacement
A	Piston area
F	Instantaneous spring force
F_{pr}	Spring preload force
F_{max}	Spring force at maximum piston displacement
$stroke$	Piston stroke
or	Actuator orientation with respect to the globally assigned positive direction. If pressure applied at port X moves the piston in positive direction, *or* equals 1. If pressure applied at port X moves the piston in negative direction, *or* equals –1.

Connection X is a hydraulic conserving port associated with the valve chamber. Connection P is a physical signal port whose output corresponds to piston displacement. Pressure applied at port X moves the piston in the positive or negative direction, depending on the value of the **Actuator orientation** parameter.

Basic Assumptions and Limitations

- No loading, such as inertia, friction, hydraulic force, and so on, is taken into account. The only force considered is a spring force.
- No flow consumption associated with the piston motion, leakage, or fluid compressibility is taken into account.

Parameters

Piston area

Effective piston area. The default value is `2e-4` m^2.

Preload force

Spring preload force. The default value is `20` N.

Full stroke force

Force necessary to move the piston to maximum stroke. The default value is `70` N.

Piston stroke

Piston stroke. The default value is `5e-3` m.

Actuator orientation

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it moves the piston in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port X moves the piston in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Actuator orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

X

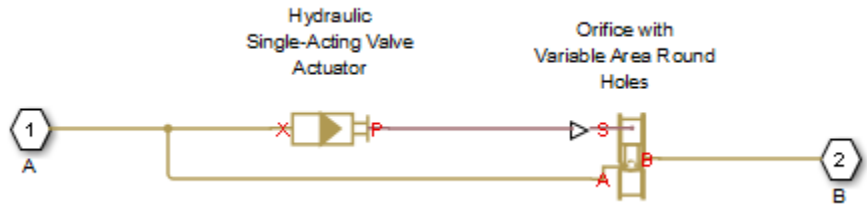
Hydraulic conserving port associated with the valve chamber.

P

Physical signal port that outputs piston displacement.

Examples

The following example shows a model of a pressure-relief valve built using the Hydraulic Single-Acting Valve Actuator and Orifice with Variable Area Round Holes blocks.



See Also

2-Position Valve Actuator | 3-Position Valve Actuator | Hydraulic Double-Acting Valve Actuator | Proportional and Servo-Valve Actuator

Introduced in R2007a

Hydraulically Operated Remote Control Valve

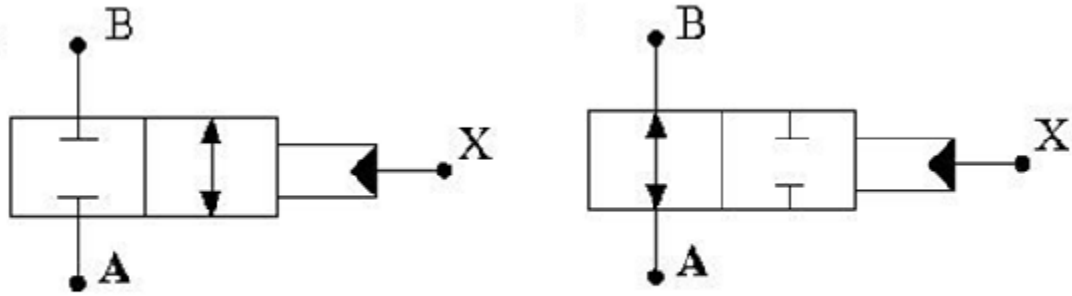
Normally closed or normally open hydraulically operated remote control valve

Library

Directional Valves

Description

The Hydraulically Operated Remote Control Valve block represents a hydraulically operated remote control valve as a data-sheet-based model, meaning that most of the model parameters are generally available in catalogs or manufacturer data sheets. Hydraulically operated remote control valves are widely used in hydraulic systems as hydraulic switches, unloading and sequencing valves. You can also use them as pressure-relief and pressure-reducing valves. The block covers both the normally closed and normally open configurations, shown in the following figure.



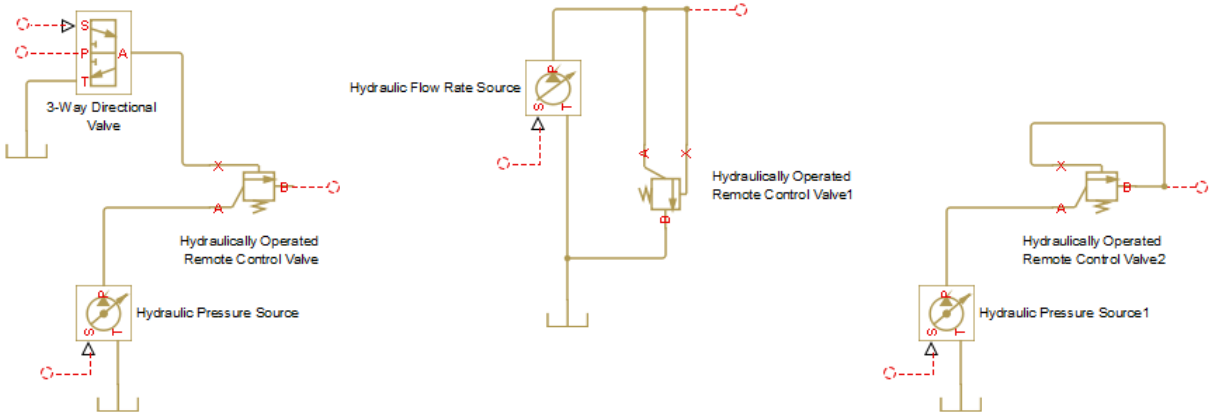
Normally closed valve configuration

Normally open valve configuration

The valve opens (closes) by the pilot pressure. The valve control member remains in its initial position as long as the pilot pressure is lower than the cracking pressure. When cracking pressure is reached, the valve control member (spool, ball, poppet, and so on) is forced off its seat and starts opening the normally closed valve, or closing the normally open valve. The control member displacement is directly proportional to pilot pressure. The member reaches its maximum displacement after the pilot pressure becomes equal or greater than the preset maximum value. The valve maximum area, cracking pressure, and maximum pressure are the key parameters of the block. These three parameters are usually provided in catalogs or data sheets.

In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Therefore, the parameter value must be greater than zero.

Schematic fragments in the next illustration show some typical valve applications: remote control valve (a), pressure-relief valve (b), and pressure-reducing valve (c).



a) Remote control valve **b) Pressure-relief valve** **c) Pressure-reducing valve**

The flow rate through the orifice is proportional to the orifice opening and the pressure differential across the orifice.

By default, the block does not include valve opening dynamics, and the valve sets its opening area directly as a function of pressure:

$$A = A(p)$$

Adding valve opening dynamics provides continuous behavior that is more physically realistic, and is particularly helpful in situations with rapid valve opening and closing. The pressure-dependent orifice passage area *A(p)* in the block equations then becomes the steady-state area, and the instantaneous orifice passage area in the flow equation is determined as follows:

$$A(t = 0) = A_{init}$$

$$\frac{dA}{dt} = \frac{A(p) - A}{\tau}$$

In either case, the flow rate through the valve is determined according to the following equations:

$$q = C_D \cdot A\sqrt{\frac{2}{\rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

For the normally closed valve, the instantaneous orifice passage area *A(p)* is computed with the equations:

$$A(p) = \begin{cases} A_{leak} & \text{for } p_p <= p_{crack} \\ A_{leak} + gain \cdot \left(p_p - p_{crack}\right) & \text{for } p_{crack} < p_p < p_{\max} \\ A_{\max} & \text{for } p_p >= p_{\max} \end{cases}$$

For the normally open valve, the equations are similar:

$$A(p) = \begin{cases} A_{\max} & \text{for } p_p <= p_{crack} \\ A_{\max} - gain \cdot \left(p_p - p_{crack}\right) & \text{for } p_{crack} < p_p < p_{\max} \\ A_{leak} & \text{for } p_p >= p_{\max} \end{cases}$$

The rest of the equations apply to both valve configurations:

$$gain = \frac{A_{\max} - A_{leak}}{p_{\max} - p_{crack}}$$

$$\Delta p = p_A - p_B,$$

where

q	Flow rate through the valve
p	Pressure differential across the valve
p_A, p_B	Gauge pressures at the block terminals
p_p	Gage pressure at the pilot terminal
C_D	Flow discharge coefficient
ρ	Fluid density
τ	Time constant for the first order response of the valve opening

t	Time
A	Instantaneous orifice passage area
$A(p)$	Pressure-dependent orifice passage area
A_{init}	Initial open area of the valve
A_{max}	Fully open valve passage area
A_{leak}	Closed valve leakage area
p_{crack}	Valve cracking pressure
p_{max}	Pilot pressure to shift the control member to its maximum
p_{cr}	Minimum pressure for turbulent flow, when the block transitions from laminar to turbulent regime

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

  $$p_{avg} = (p_A + p_B)/2$$

  where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

  $$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Orifice hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

Connections A and B are hydraulic conserving ports associated with the inlet and the outlet of the valve. Connection X is the pilot port, which is a hydraulic conserving port that provides the pilot pressure. The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as $\Delta p = p_A - p_B$, .

Basic Assumptions and Limitations

- Control member displacement is linearly proportional to pilot pressure.
- No flow consumption is associated with the pilot chamber.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.

Parameters

Valve type

Select the valve configuration: `Normally closed valve` or `Normally open valve`. The default is `Normally closed valve`.

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Cracking pressure

Pressure level at which the valve control member is forced off its seat and starts to either open or close the valve, depending on the valve type. The default value is `3e4` Pa.

Maximum control member displacement pressure

Pilot pressure at which the valve control member shifts to its maximum displacement and remains there until the pressure falls below this level. Its value must be higher than the cracking pressure. The default value is `1.2e5` Pa.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Opening dynamics

Select one of the following options:

- `Do not include valve opening dynamics` — The valve sets its orifice passage area directly as a function of pressure. If the area changes instantaneously, so does the flow equation. This is the default.
- `Include valve opening dynamics` — Provide continuous behavior that is more physically realistic, by adding a first-order lag during valve opening and

closing. Use this option in hydraulic simulations with the local solver for real-time simulation. This option is also helpful if you are interested in valve opening dynamics in variable step simulations.

Opening time constant

The time constant for the first order response of the valve opening. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `0.1` s.

Initial area

The initial opening area of the valve. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `1e-12` m^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Valve type**
- **Opening dynamics**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port that acts as the control port and provides the pilot pressure.

See Also

Pressure Reducing Valve | Pressure Relief Valve

Introduced in R2012b

Interface (TL-IL)

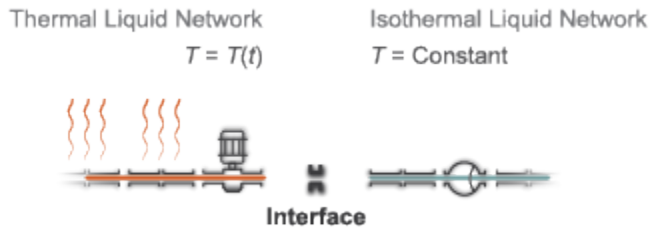
Junction between thermal liquid and isothermal liquid networks
Library: Simscape / Fluids / Fluid Network Interfaces

Description

The Interface (TL-IL) block represents a cross-domain junction between thermal liquid and hydraulic (isothermal liquid) networks. The interface behaves as a moving membrane, allowing fluid networks to exchange mass and momentum while preventing the respective fluids from mixing.

Use this block as a modeling convenience when thermal effects matter in parts of a system but not in others. You can model the thermal effects using Thermal Liquid blocks and interface the resulting physical network with a Hydraulic model based on a constant-temperature assumption.

Pressure and mass flow rate are held constant across the interface. Temperature is free to fluctuate on the thermal liquid side of the interface but fixed at a constant value on the hydraulic side. Fluid properties are network-specific and do not extend across the interface.



Mass Flow Rate

Mass is conserved at the interface. The interface wall is assumed rigid and the fluid dynamic compressibility ignored. With these assumptions, the mass accumulation rate

inside the interface reduces to zero. Noting that the mass flow rate in the Hydraulic domain is computed from volumetric flow rate data:

$$\dot{m}_A + q_{\mathrm{B}}\rho_{\mathrm{B}} = 0,$$

where:

- $\dot{m}_A$ is the mass flow rate into the interface through port A.
- q_{B} is the volumetric flow rate into the interface through port B.
- ρ_{B} is the hydraulic (isothermal liquid) density at port B.

Pressure

Momentum is conserved at the interface. Viscous friction, fluid dynamic compressibility, and fluid inertia are ignored. With these assumptions, the momentum balance reduces to an equality statement between the pressures at port A and port B. Noting that pressure in the Thermal Liquid domain is absolute while that in the Hydraulic domain is gauge:

$$p_{\mathrm{A}} = p_{\mathrm{B}} + 1\mathrm{atm},$$

where:

- p_{A} is the thermal liquid absolute pressure at port A.
- p_{B} is the hydraulic (isothermal liquid) gauge pressure at port B.

Temperature

The Simscape Hydraulic domain has no concept of temperature. The block computes the heat transfer rate between the thermal liquid port and the interface interior only. The temperature inside the interface is assumed to be held constant at a specified value:

$$T_{\mathrm{I}} = \mathrm{T}_{\mathrm{Hyd}},$$

where:

- T_{I} is the thermal liquid temperature inside the interface.
- T_{Hyd} is the **Hydraulic fluid temperature** block parameter.

The temperature at the thermal liquid port is a function of the convective and conductive heat flow rates inside the interface. The conductive heat flow rate becomes significant only at mass flow rate approaching zero.

Assumptions

- Viscous friction, fluid inertia, and dynamic compressibility are assumed negligible.
- The interface walls are assumed rigid.

Ports

Conserving

`A` — Thermal liquid inlet
thermal liquid

Thermal liquid conserving port representing a passage point through the interface. Connect this port to a block network built on the Simscape Thermal Liquid domain.

`B` — Isothermal liquid inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing a passage point through the interface. Connect this port to a block network built on the Simscape Hydraulic domain.

Parameters

`Hydraulic fluid temperature` — Temperature on the hydraulic (isothermal liquid) side of the interface
293.15 K (default)

Temperature in the hydraulic (isothermal liquid) network. The block uses this parameter to calculate the heat transfer rate across the interface. If the isothermal liquid network contains a Hydraulic Fluid block, ensure that the specified temperature matches the **System temperature (C)** parameter in that block.

`Cross-sectional area at port A` — Area normal to the direction of flow at the thermal liquid inlet
0.01 m^2 (default)

Area normal to the direction of flow at port A. The block uses this parameter to calculate the heat transfer rate across the interface. Ensure that the specified area matches the cross-sectional areas of the adjacent component inlets.

See Also

E-NTU Heat Transfer | Heat Exchanger Interface (TL)

Introduced in R2016b

Jet Pump

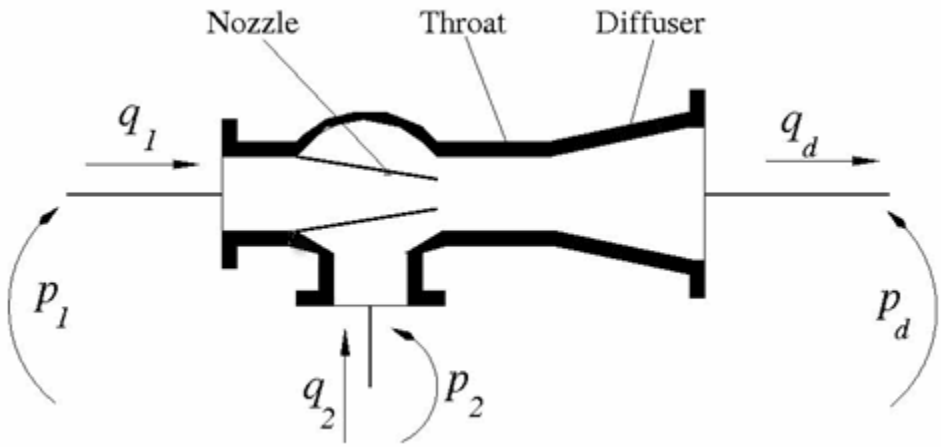
Jet liquid-liquid pump

Library

Pumps and Motors

Description

The Jet Pump block represents a jet liquid-liquid pump consisting of a nozzle, throat, and diffuser, as shown in the following illustration.



The model is based on the following equations, described in [1 on page 1-785]:

$$q_1 = \frac{A_n}{\sqrt{1+K_n}}\sqrt{\frac{2}{\rho}(p_1 - p_0)} \tag{1-8}$$

$$q_2 = \frac{A_n \cdot c}{\sqrt{1+K_{en}}}\sqrt{\frac{2}{\rho}(p_2 - p_0)} \tag{1-9}$$

$$p_d - p_0 = Zb^2\left(\frac{2}{b} + \frac{2}{1-b}M^2 - (1+M)^2 \cdot \left(1 + K_{th} + K_{di} + \alpha^2\right)\right) \tag{1-10}$$

$$b = \frac{A_n}{A_{th}}$$

$$c = \frac{1-b}{b}$$

$$Z = \rho\frac{V_n^2}{2} = \rho\frac{q_1^2}{2A_n^2}$$

$$M = \frac{q_2}{q_1}$$

where

q_1	Primary flow rate pumped through the nozzle
q_2	Secondary flow rate
q_d	Output flow rate
p_1	Pressure at the nozzle inlet
p_2	Pressure at the secondary flow rate inlet
p_0	Pressure at the throat inlet
p_d	Pressure at the pump outlet
A_n	Nozzle area

A_{th}	Throat area
a	Diffuser area ratio, A_{th} / A_d
A_d	Diffuser outlet area
K_n	Nozzle hydraulic loss coefficient
K_{en}	Throat entry hydraulic loss coefficient
K_{th}	Throat hydraulic loss coefficient
K_{di}	Diffuser hydraulic loss coefficient
ρ	Fluid density

“Equation 1-8” describes the nozzle, “Equation 1-9” – throat entry, and “Equation 1-10” – the combination of the throat and the diffuser. The equations correspond to a standard configuration of the pump, where all the longitudinal dimensions conform to established, empirically determined values. For more details, see [1 on page 1-785].

The pump parameters are closely related to each other, and the methodology described in [1 on page 1-785] is recommended to determine their initial values.

Basic Assumptions and Limitations

- The model is based on the one-dimensional theory.
- The primary and secondary flows enter the mixing throat with uniform velocity distribution, and the mixed flow leaves the diffuser with uniform velocity distribution.
- The fluid in the primary and secondary flows is the same.
- The fluid is assumed to be incompressible and containing no gas.

Parameters

Nozzle area

Cross-sectional area of the nozzle. The parameter must be greater than zero. The default value is 1 cm^2.

Throat area

Cross-sectional area of the throat. The throat area is usually two to four times larger than the nozzle area. The default value is 4 cm^2.

Diffuser inlet/outlet area ratio

The ratio between the inlet and outlet diffuser areas. For a standard pump with a 5° – 7° included-angle diffuser, the ratio is close to 0.2. The parameter must be greater or equal to zero. The default value is `0.224`.

Nozzle loss coefficient

The hydraulic friction loss coefficient in the nozzle. The parameter must be greater than zero. The default value is `0.05`.

Throat entry loss coefficient

The hydraulic friction loss coefficient in the throat entry. The parameter must be greater than zero. The default value is `0.005`.

Throat loss coefficient

The hydraulic friction loss coefficient in the throat. The parameter must be greater than zero. The default value is `0.1`.

Diffuser loss coefficient

The hydraulic friction loss coefficient in the diffuser. The parameter must be greater than zero. The default value is `0.1`.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the nozzle entry (primary flow entry).

`S`

Hydraulic conserving port associated with the pump suction (secondary flow entry).

P

Hydraulic conserving port associated with the pump outlet.

N

Internal nonvisible hydraulic conserving port associated with the throat entry section of the pump. You can view the variables associated with the port by logging simulation data. For more information, see “Data Logging” (Simscape).

Examples

The Well with Jet Pump example represents a well jet pump installation, consisting of a surface-mounted centrifugal pump and a jet pump installed in the well below water level.

References

[1] I.J. Karassic, J.P. Messina, P. Cooper, C.C. Heald, *Pump Handbook*, Fourth edition, McGraw-Hill, NY, 2008

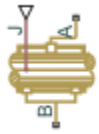
See Also

Centrifugal Pump | Fixed-Displacement Pump | Variable-Displacement Pressure-Compensated Pump | Variable-Displacement Pump

Introduced in R2010b

Journal Bearing Pressure-Fed

Hydraulic portion of pressure-fed journal bearing

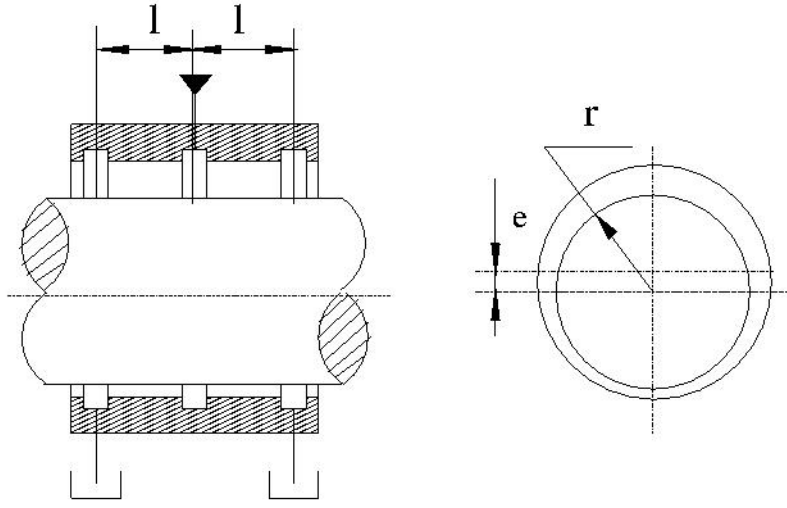


Library

Orifices

Description

The Journal Bearing Pressure-Fed block simulates the hydraulic portion of a pressure-fed journal bearing, shown in the following illustration.



The lubricant under pressure p is pumped into the circumferential groove at the center of the bearing. The groove divides the bearing into two half-bearings. The lubricant exits through the end grooves located at a distance l from the central groove. The model is intended to be used in lubrication system simulation to assess the flow consumption through the pressure-fed journal bearing. The flow regime is assumed to be laminar due to very small clearances between the journal and the bushing.

The flow rate is computed using the Hagen-Poiseuille equation (see [1] on page 1-789):

$$q = \frac{\pi p r c^3}{3\mu l}\left(1 + 1.5\varepsilon^2\right)$$

where

q	Volumetric flow rate
p	Pressure differential across the bearing
r	Journal radius
c	Radial clearance at neutral position
μ	Dynamic viscosity
l	Length of the half-bearing
ε	Relative eccentricity, $\varepsilon = e / r$
e	Eccentricity or journal deflection from the central position

The journal radial displacement, which controls the bearing eccentricity, is imported through the physical signal port J. Connections A and B are hydraulic conserving ports associated with the bearing inlet and outlet, respectively. The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as $\Delta p = p_A - p_B$, . Positive signal at the physical signal port J increases the eccentricity and is limited to the radial clearance of the bearing.

Basic Assumptions and Limitations

The flow regime is assumed to be laminar due to small clearances.

Parameters

Journal radius

The radius of the journal. The parameter must be positive. The default value is `0.05` m.

Radial clearance

The radial clearance between the journal and the bushing at neutral position. The parameter must be positive. The default value is `2e-4` m.

Bearing half-length

The length of the half-bearing, that is, the distance between each of the end grooves and the central groove. The parameter must be positive. The default value is `0.025` m.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the bearing inlet.

`B`

Hydraulic conserving port associated with the bearing outlet.

`J`

Physical signal port that controls the journal deflection.

References

[1] Shigley, J., C. Mischke, and R. Budynas. *Mechanical Engineering Design*. New York: McGraw-Hill, 2004.

Introduced in R2012b

Local Resistance

Hydraulic resistance specified by loss coefficient

Library

Local Hydraulic Resistances

Description

The Local Resistance block represents a generic local hydraulic resistance, such as a bend, elbow, fitting, filter, local change in the flow cross section, and so on. The pressure loss caused by resistance is computed based on the pressure loss coefficient, which is usually provided in catalogs, data sheets, or hydraulic textbooks. The pressure loss coefficient can be specified either as a constant, or by a table, in which it is tabulated versus Reynolds number.

The pressure drop between port **A** and port **B** is:

$$\Delta p = p_A - p_B,$$

where:

- p is the pressure drop.
- p_A is the gauge pressure at port **A**.
- p_B is the gauge pressure at port **B**.

If the **Model parameterization** parameter is set to `By semi-empirical formulas`, the pressure drop is related to the volumetric flow rate by the expression:

$$q = A\sqrt{\frac{2}{K \cdot \rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

where:

- q is the volumetric flow rate.
- A is the flow area.
- K is the pressure loss coefficient of the flow resistance.
- ρ is the fluid density.
- p_{Cr} is the minimum pressure for turbulent flow.

If the **Model parameterization** parameter is set to `By loss coefficient vs. Re table`, the pressure drop is related to the volumetric flow rate by the expression:

$$p = K(\mathrm{Re})\rho\frac{q|q|}{2A^2}$$

where K is now a function of the Reynolds number (Re). Its value is specified in the block dialog box in tabulated form against the Reynolds number:

$$\mathrm{Re} = \frac{q \cdot D_H}{A \cdot \nu}$$

where:

- D_H is the hydraulic diameter of the flow resistance:

  $$D_H = \sqrt{\frac{4A}{\pi}}$$

- ν is the kinematic viscosity.

For a constant pressure loss coefficient, the minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

  $$p_{avg} = (p_A + p_B)/2$$

  where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = K\frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{D_H}\right)^2$$

where

Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

The block provides two parameterizations:

- `By semi-empirical formulas` — The pressure loss coefficient is assumed to be constant for a specific flow direction. The flow regime can be either laminar or turbulent, depending on the Reynolds number.
- `By loss coefficient vs. Re table` — The pressure loss coefficient is specified as a function of the Reynolds number. The flow regime is assumed to be turbulent at all times. You must ensure that the loss coefficient data corresponds to this flow regime.

The resistance can be symmetrical or asymmetrical. In symmetrical resistances, the pressure loss practically does not depend on flow direction and one value of the coefficient is used for both the direct and reverse flow. For asymmetrical resistances, the separate coefficients are provided for each flow direction. If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

Connections A and B are conserving hydraulic ports associated with the block inlet and outlet, respectively.

The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $\Delta p = p_A - p_B$, .

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- If you select parameterization by the table-specified relationship *K=f(Re)*, the flow is assumed to be completely turbulent.

Parameters

- “Parameters Tab” on page 1-793
- “Variables Tab” on page 1-795

Parameters Tab

Resistance area

The smallest passage area. The default value is `1e-4` m^2.

Model parameterization

Select one of the following methods for specifying the pressure loss coefficient:

- `By semi-empirical formulas` — Provide a scalar value for the pressure loss coefficient. For asymmetrical resistances, you have to provide separate coefficients for direct and reverse flow. This is the default method.
- `By loss coefficient vs. Re table` — Provide tabulated data of loss coefficients and corresponding Reynolds numbers. The loss coefficient is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods. For asymmetrical resistances, the table must cover both the positive and the negative flow regions.

Pressure loss coefficient for direct flow

Loss coefficient for the direct flow (flowing from A to B). For simple ideal configurations, the value of the coefficient can be determined analytically, but in most cases its value is determined empirically and provided in textbooks and data sheets (for example, see [1] on page 1-797). The default value is `2`. This parameter is used if **Model parameterization** is set to `By semi-empirical formulas`.

Pressure loss coefficient for reverse flow

Loss coefficient for the reverse flow (flowing from B to A). The parameter is similar to the loss coefficient for the direct flow and must be set to the same value if the

resistance is symmetrical. The default value is 2. This parameter is used if **Model parameterization** is set to `By semi-empirical formulas`.

Laminar transition specification

If **Model parameterization** is set to `By semi-empirical formulas`, select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `150`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Reynolds number vector

Specify the vector of input values for Reynolds numbers as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values are `[-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]`. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Loss coefficient vector

Specify the vector of the loss coefficient values as a one-dimensional array. The loss coefficient vector must be of the same size as the Reynolds numbers vector. The default values are `[0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]`. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Variables Tab

Flow rate

Volumetric flow rate through the local resistance at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Pressure differential

Pressure differential between the local resistance ports at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**
- **Extrapolation method**
- **Laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the resistance inlet.

B

Hydraulic conserving port associated with the resistance outlet.

References

[1] Idelchik, I.E., *Handbook of Hydraulic Resistance*, CRC Begell House, 1994

See Also

Elbow | Gradual Area Change | Pipe Bend | Sudden Area Change | T-junction

Introduced in R2006a

Needle Valve

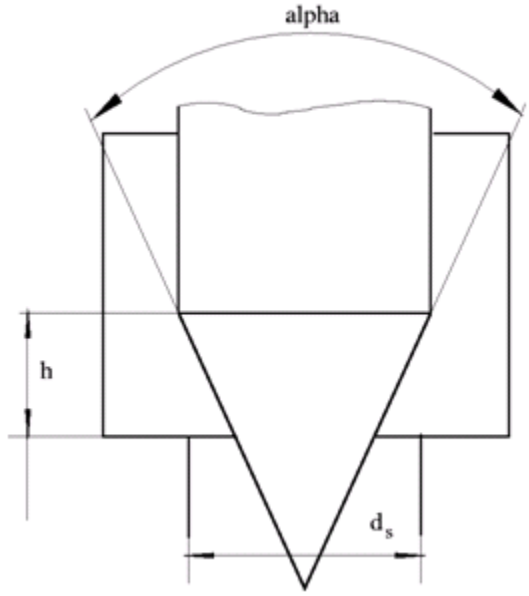
Hydraulic needle valve

Library

Flow Control Valves

Description

The Needle Valve block models a variable orifice created by a conical needle and a round sharp-edged orifice in thin material.



The flow rate through the valve is proportional to the valve opening and to the pressure differential across the valve. The flow rate is determined according to the following equations:

$$q = C_D \cdot A(h)\sqrt{\frac{2}{\rho}}\frac{\Delta p}{\left(\Delta p^2 + p_{\mathrm{Cr}}^2\right)^{1/4}},$$

$$\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}},$$

$$h = x_0 + x$$

$$h_{\max} = d_s \frac{1 - \sqrt{1 - \cos\frac{\alpha}{2}}}{\sin\alpha}$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h <= 0 \\ \pi \cdot \left(d_s - h \cdot \sin\frac{\alpha}{2} \cdot \cos\frac{\alpha}{2} \right) \cdot h \cdot \sin\frac{\alpha}{2} + A_{leak} & \text{for } 0 < h < h_{\max} \\ A_{\max} + A_{leak} & \text{for } h >= h_{\max} \end{cases}$$

$$A_{\max} = \frac{\pi d_s^2}{4}$$

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Needle displacement from initial position
h	Valve opening
h_{max}	Maximum needle stroke
d_s	Orifice diameter
α	Needle angle
ρ	Fluid density
A_{leak}	Closed valve leakage area
A_{max}	Maximum valve open area
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

$$p_{avg} = (p_A + p_B)/2$$

where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as

$\Delta p = p_A - p_B$, . Positive signal at the physical signal port S opens the valve.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- The flow passage area is assumed to be equal to the frustum side surface area.

Parameters

Valve orifice diameter

The diameter of the orifice of the valve. The default value is `0.005` m.

Needle cone angle

The angle of the valve conical needle. The parameter value must be in the range between 0 and 180 degrees. The default value is `90` degrees.

Initial opening

The initial opening of the valve. You can specify both positive and negative values. The default value is `0`.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.65`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `10`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion

of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12 m^2`.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the valve inlet.

`B`

Hydraulic conserving port associated with the valve outlet.

`S`

Physical signal port to control spool displacement.

See Also

Ball Valve | Gate Valve | Poppet Valve | Pressure-Compensated Flow Control Valve

Introduced in R2006a

Orifice ISO 6358 (G)

Flow restriction of fixed area modeled per ISO 6358
Library: Simscape / Fluids / Gas / Valves & Orifices

Description

The Orifice ISO 6358 (G) block models the pressure loss incurred in a gas network due to a purely resistive element of fixed size—such as a flow restriction, an orifice, or a valve—using the methods outlined in the ISO 6358 standard. These methods are widely used in industry in the measurement and reporting of gas flow characteristics. The availability of data on the coefficients of the ISO formulas makes the ISO parameterizations useful when component geometries are unavailable or cumbersome to specify.

Orifice Parameterizations

The default orifice parameterization is based on the most recommended of the ISO 6358 methods: one based on the *sonic conductance* of the resistive element at steady state. The sonic conductance measures the ease with which a gas can flow when *choked*, a condition in which the flow velocity is at its theoretical maximum (the local speed of sound). Choking occurs when the ratio between downstream and upstream pressures reaches a critical value known as the *critical pressure ratio*

The remaining parameterizations are formulated in terms of alternative measures of flow capacity: the flow coefficient (in either of its forms, C_v or K_v) or the size of the flow restriction. The flow coefficient measures the ease with which a gas can flow when driven by a certain pressure differential. The definition of C_v differs from that of K_v in the standard pressure and temperature established in its measurement and in the physical units used in its expression:

- C_v is measured at a generally accepted temperature of `60°F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Orifice parameterization** block parameter is set to `Cv coefficient (USCS)`.

- K_v is measured at a generally accepted temperature of `15°C` and pressure drop of `1 bar`; it is expressed in metric units of `m^3/h`. This is the flow coefficient used in the model when the **Orifice parameterization** block parameter is set to `Kv coefficient (SI)`.

Mass Balance

The volume of fluid inside the resistive element, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of fluid can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through the port indicated by the subscript (**A** or **B**).

Momentum Balance

The causes of the pressure losses incurred in the passages of the resistive element are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. It is this cumulative effect that the sonic conductance in the default orifice parameterization captures in a model. If a different parameterization is selected, the coefficients on which it is based are converted into the parameters of the default parameterization; the mass flow rate calculation is then carried out as described in Sonic Conductance Parameterization.

In a choked flow, the mass flow rate through the resistive element is calculated as:

$$\dot{m}_{\mathrm{ch}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}},$$

where:

- C is the sonic conductance inside the resistive element.

- ρ is the gas density, here at standard conditions (subscript `0`, `1.185 kg/m^3`).
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (subscript `in`) or at standard conditions (subscript `0`, `293.15 K`).

In a subsonic and turbulent flow, the mass flow rate calculation becomes:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{p_{\mathrm{r}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m ,$$

where:

- p_{r} is the ratio between downstream pressure (p_{out}) and upstream pressure (p_{in}) (each measured against absolute zero):

  $$p_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the gas flow first begins to choke.
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows.

In a subsonic and laminar flow, the mass flow rate calculation changes to:

$$\dot{m}_{\mathrm{lam}} = C\rho_0 \left(\frac{p_{\mathrm{out}} - p_{\mathrm{in}}}{1 - b_{\mathrm{lam}}} \right) \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{b_{\mathrm{lam}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m ,$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes. Combining the calculations for the three flow regimes into a piecewise function gives across all pressure ratios:

$$\dot{m} = \begin{cases} \dot{m}_{\mathrm{lam}}, & \text{if } b_{\mathrm{lam}} \le p_{\mathrm{r}} < 1 \\ \dot{m}_{\mathrm{tur}}, & \text{if } b_{\mathrm{cr}} \le p_{\mathrm{r}} < b_{\mathrm{lam}} \\ \dot{m}_{\mathrm{ch}}, & \text{if } p_{\mathrm{r}} < b_{\mathrm{cr}} \end{cases} ,$$

If the orifice parameterization is set to `Cv coefficient (USCS)`, the parameters of the mass flow rate calculation are set as follows:

- **Sonic conductance**: C = 4E-8 * C_v m^3/(s*Pa)
- **Critical pressure ratio**: b_{cr} = 0.3
- **Subsonic index**: m = 0.5

If the `Kv coefficient (SI)` parameterization is used:

- **Sonic conductance**: C = 4.78E-8 * K_v m^3/(s*Pa)
- **Critical pressure ratio**: b_{cr} = 0.3
- **Subsonic index**: m = 0.5

For the `Restriction area` parameterization:

- **Sonic conductance**: C = 0.128 * 4 S_R/π L/(s*bar), where S is the flow area in the resistive element (subscript R).
- **Critical pressure ratio**: b_{cr} = 0.41 + 0.272 (S_R/S_P)^0.25
- **Subsonic index**: m = `0.5`

Energy Balance

The resistive element is modeled as an adiabatic component. No heat exchange can occur between the fluid and the wall that surrounds it. No work is done on or by the fluid as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_A + \phi_B = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Flow passage
gas

Opening through which the flow can enter or exit the flow resistance. Which of the ports serves as inlet and which as outlet depends on the direction of flow.

B — Flow passage
gas

Opening through which the flow can enter or exit the flow resistance. Which of the ports serves as inlet and which as outlet depends on the direction of flow.

Parameters

`Orifice parameterization` — Parameterization by which to specify the flow characteristics of the orifice
`Sonic conductance` (default) | `Cv coefficient (USCS)` `Kv coefficient (S)` `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the `Sonic conductance` parameterization. If a different parameterization is selected, the coefficients that characterize it are converted into sonic conductance, critical pressure ratio, and subsonic index.

`Sonic conductance` — Measure of flow rate capacity in the choked flow regime
`1.6 l/(s*bar)` (default) | positive scalar in units of volume/(time*pressure)

Ratio, measured at the onset of choking, of the mass flow rate through the resistive element to the product of the upstream pressure and mass density at standard conditions (defined in ISO8778). This parameter determines the maximum flow rate allowed at a given upstream pressure.

`Critical pressure ratio` — Ratio of downstream and upstream pressures at which the flow first chokes
`0.3` (default) | positive unitless scalar

Pressure ratio at which flow first begins to choke and the flow velocity reaches its maximum, given by the local speed of sound. The pressure ratio is the fraction of the absolute pressure downstream of the resistive element over the absolute pressure upstream of the same component.

`Subsonic index` — Exponent used to more accurately characterize mass flow in the subsonic flow regime
`0.5` (default) | positive unitless scalar

Exponent used to more accurately calculate the mass flow rate in the subsonic flow regime as described in ISO 6358.

`Cv coefficient (USCS)` — Flow coefficient expressed in US customary units
`0.4` (default) | positive unitless scalar

Flow coefficient expressed in US customary units of ft^3/min as defined in NFPA T3.21.3. This parameter measures the ease with which the gas traverses the resistive element when driven by a pressure differential. See the block description for the correspondence between these parameters.

`Kv coefficient (SI)` — Flow coefficient expressed in SI units
`0.3` (default) | positive unitless scalar

Flow coefficient expressed in SI units of L/min. This parameter measures the ease with which the gas traverses the resistive element when driven by a pressure differential. See the block description for the correspondence between these parameters.

`Restriction area` — Flow area of the local restriction
`1e-5 m` (default) | positive scalar in units of area

Area normal to the direction of flow at the point of shortest aperture. The restriction area is converted into an equivalent sonic conductance and critical pressure ratio for use in calculations of mass flow rate. See the block description for detail on the conversion.

`Cross-sectional area at ports A and B` — Flow area at the gas ports
`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the resistive element connects.

`Laminar flow pressure ratio` — Pressure ratio at which flow transitions between laminar and turbulent regimes
`0.999` (default) | positive unitless scalar

Pressure ratio at which flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the resistive element over the absolute pressure upstream of the same component. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Temperature established in the measurement of sonic conductance
`293.15 K` (default) | positive scalar in units of temperature

Absolute temperature established in the measurement of sonic conductance as defined in ISO 8778.

`Reference density` — Density established in the measurement of sonic conductance
`1.185 kg/m^3` (default) | positive scalar in units of mass/volume

Mass density at the standard conditions established in the measurement of sonic conductance as defined in ISO 8778.

References

[1] P. Beater, *Pneumatic Drives*, Springer-Verlag Berlin Heidelberg, 2007.

See Also

Variable Orifice ISO 6358 (G)

Introduced in R2018a

Variable Orifice ISO 6358 (G)

Flow restriction of variable area modeled per ISO 6358
Library: Simscape / Fluids / Gas / Valves & Orifices

Description

The Variable Orifice ISO 6358 (G) block models the pressure loss incurred in a gas network due to a purely resistive element of variable size—such as a controlled flow restriction, orifice, or valve—using the methods outlined in the ISO 6358 standard. These methods are used in industry in the measurement and reporting of gas flow characteristics. The availability of data on the coefficients of the ISO formulas makes the ISO parameterizations useful when component geometries are unavailable or cumbersome to specify.

Orifice Parameterizations

The default orifice parameterization is based on the most recommended of the ISO 6358 methods: one based on the *sonic conductance* of the resistive element at steady state. The sonic conductance measures the ease with which a gas can flow when *choked*, a condition in which the flow velocity is at its theoretical maximum (the local speed of sound). Choking occurs when the ratio between downstream and upstream pressures reaches a critical value known as the *critical pressure ratio*

The remaining parameterizations are formulated in terms of alternative measures of flow capacity: the flow coefficient (in either of its forms, C_v or K_v) or the size of the flow restriction. The flow coefficient measures the ease with which a gas can flow when driven by a certain pressure differential. The definition of C_v differs from that of K_v in the standard pressure and temperature established in its measurement and in the physical units used in its expression:

- C_v is measured at a generally accepted temperature of `60°F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Orifice parameterization** block parameter is set to `Cv coefficient (USCS)`.

- K_v is measured at a generally accepted temperature of `15°C` and pressure drop of `1 bar`; it is expressed in metric units of `m^3/h`. This is the flow coefficient used in the model when the **Orifice parameterization** block parameter is set to `Kv coefficient (SI)`.

Two values are required for the chosen measure of flow capacity (that for which the orifice parameterization is named): a *maximum* and a *minimum*. The maximum corresponds to a valve open to full capacity; this is the value for which the coefficient data are frequently reported in valve data sheets. The minimum corresponds to a valve closed tight, when only leakage flow remains, if any at all. This lower bound serves primarily to ensure the numerical robustness of the model. Its exact value is less important than its being a (generally very small) number greater than zero.

Opening Parameterizations

The sonic conductance and (in certain settings) the critical pressure ratio are determined during simulation from the input at port **L**. This input is the *control signal* and it is, in some valves, associated with *stroke* or *lift percent*. The control signal can range in value from `0` to `1`. If a lesser or greater value is specified, it is adjusted to the nearest of the two limits. In other words, the signal is *saturated* at `0` and `1`.

If the orifice parameterization is changed from its default of `Sonic conductance`, the sonic conductance and critical pressure ratio are determined as linear functions of the chosen measure of flow capacity. This alternative measure is in turn obtained from the control signal. Calculations of mass flow rate are carried out as before, using the equations described in ``Sonic Conductance Parameterization''.

The conversion from a control signal to the chosen measure of flow capacity depends on the *opening* parameterization selected in the block. Flow is always maximally restricted when the control signal is `0` and minimally so when the control signal is `1`. However, in between, the flow rate achieved within the resistive element depends on whether the opening parameterization is linear or based on tabulated data:

- `Linear` — The measure of flow capacity (sonic conductance, Cv coefficient, other) is proportional to the control signal at port **L**. The two vary in tandem until the control signal either drops below `0` (flow is maximally restricted) or rises above `1` (flow is minimally restricted). As the control signal rises from `0` to `1`, the measure of flow capacity scales from its specified minimum to its specified maximum.

In the conversion to the parameters of the sonic conductance parameterization, both the critical pressure ratio and the subsonic index are treated as constants, each independent of the control signal.

- `Tabulated data` — The measure of flow capacity is a tabulated function of the control signal at port **L**. This function is based on a one-way lookup table with the control signal corresponding to the abscissa and the measure of flow capacity to the ordinate. The tabulated data must be specified such that the measure of flow capacity increases monotonically with the control signal.

  In the conversion to the parameters of the sonic conductance parameterization, the critical pressure ratio is treated as a function of the control signal while the subsonic index is treated as a constant.

Mass Balance

The volume of fluid inside the resistive element, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of fluid can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through the port indicated by the subscript (**A** or **B**).

Momentum Balance

The causes of the pressure losses incurred in the passages of the resistive element are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. It is this cumulative effect that the sonic conductance in the default orifice parameterization captures in a model. If a different parameterization is selected, the coefficients on which it is based are converted into the parameters of the default parameterization; the mass flow rate calculation is then carried out as described in Sonic Conductance Parameterization.

In a choked flow, the mass flow rate through the resistive element is calculated as:

$$\dot{m}_{ch} = C\rho_0 p_{in}\sqrt{\frac{T_0}{T_{in}}},$$

where:

- C is the sonic conductance inside the resistive element.
- ρ is the gas density, here at standard conditions (subscript `0`, `1.185 kg/m^3`).
- p is the absolute gas pressure, here corresponding to the inlet (subscript `in`).
- T is the gas temperature at the inlet (subscript `in`) or at standard conditions (`0`, `293.15 K`).

In a subsonic and turbulent flow, the mass flow rate calculation becomes:

$$\dot{m}_{tur} = C\rho_0 p_{in}\sqrt{\frac{T_0}{T_{in}}}\left[1-\left(\frac{p_r - b_{cr}}{1-b_{cr}}\right)^2\right]^m,$$

where:

- p_r is the ratio between downstream pressure (p_{out}) and upstream pressure (p_{in}) (each measured against absolute zero):

  $$p_r = \frac{p_{out}}{p_{in}}$$

- b_{cr} is the critical pressure ratio at which the gas flow first begins to choke.
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows.

In a subsonic and laminar flow, the mass flow rate calculation changes to:

$$\dot{m}_{lam} = C\rho_0\left(\frac{p_{out} - p_{in}}{1-b_{lam}}\right)\sqrt{\frac{T_0}{T_{in}}}\left[1-\left(\frac{b_{lam} - b_{cr}}{1-b_{cr}}\right)^2\right]^m,$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes. Combining the calculations for the three flow regimes into a piecewise function gives across all pressure ratios:

$$\dot{m} = \begin{cases} \dot{m}_{\mathrm{lam}}, & \text{if } b_{\mathrm{lam}} \le p_{\mathrm{r}} < 1 \\ \dot{m}_{\mathrm{tur}}, & \text{if } b_{\mathrm{cr}} \le p_{\mathrm{r}} < b_{\mathrm{lam}} \\ \dot{m}_{\mathrm{ch}}, & \text{if } p_{\mathrm{r}} < b_{\mathrm{cr}} \end{cases},$$

If the orifice parameterization is set to `Cv coefficient (USCS)`, the parameters of the mass flow rate calculation are set as follows:

- **Sonic conductance**: C = 4E-8 * C_{v} m^3/(s*Pa)
- **Critical pressure ratio**: b_{cr} = 0.3
- **Subsonic index**: m = 0.5

If the `Kv coefficient (SI)` parameterization is used:

- **Sonic conductance**: C = 4.78E-8 * K_{v} m^3/(s*Pa)
- **Critical pressure ratio**: b_{cr} = 0.3
- **Subsonic index**: m = 0.5

For the `Restriction area` parameterization:

- **Sonic conductance**: C = 0.128 * 4 S_{R}/π L/(s*bar), where S is the flow area in the resistive element (subscript R).
- **Critical pressure ratio**: b_{cr} = 0.41 + 0.272 ($S_{\mathrm{R}}/S_{\mathrm{P}}$)^0.25
- **Subsonic index**: m = `0.5`

Energy Balance

The resistive element is modeled as an adiabatic component. No heat exchange can occur between the fluid and the wall that surrounds it. No work is done on or by the fluid as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Input

`L` — Orifice opening, `unitless`

physical signal

Control signal by means of which to specify the opening (in some valves associated with *stroke* or *lift percent*) of the orifice. The orifice is fully closed at a value of `0` and fully open at a value of `1`.

Conserving

`A` — Flow passage

gas

Opening through which the flow can enter or exit the flow resistance. Which of the ports serves as inlet and which as outlet depends on the direction of flow.

`B` — Flow passage

gas

Opening through which the flow can enter or exit the flow resistance. Which of the ports serves as inlet and which as outlet depends on the direction of flow.

Parameters

`Orifice parameterization` — Parameterization by which to specify the flow characteristics of the orifice

`Sonic conductance` (default) | `Cv coefficient (USCS)` `Kv coefficient (S)` `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the `Sonic conductance` parameterization. If a different parameterization is selected, the coefficients that characterize it are converted into sonic conductance, critical pressure ratio, and subsonic index.

`Opening parameterization` — Method of conversion from control signal to measure of flow capacity
`Linear` (default) | `Tabulated data`

Method by which to convert the control signal specified at port **L** to the chosen measure of flow capacity (sonic conductance, either of the flow coefficients, or restriction area). See the block description for more detail on the opening parameterizations.

`Sonic conductance at maximum flow` — Sonic conductance corresponding to a maximally open component
`1.6 L/s/bar` (default) | positive scalar in units of volume/(time*pressure)

Value of the sonic conductance when the control signal specified at port **L** is `1`. The cross-sectional area available for flow is then at a maximum. During simulation the sonic conductance at intermediate control signals (those valued between `0` and `1`) is set by linear interpolation between this value and that of the **Sonic conductance at leakage flow** parameter.

`Sonic conductance at leakage flow` — Sonic conductance corresponding to a maximally closed component
`1e-5 L/s/bar` (default) | positive scalar in units of volume/(time*pressure)

Value of the sonic conductance when the control signal specified at port **L** is `0`. The cross-sectional area available for flow is then at a minimum, with only a negligible leakage flow remaining between the ports. The primary purpose of this parameter is to ensure the numerical robustness of the model during simulation. Its exact value is less important than its being a very small number greater than zero.

During simulation the sonic conductance at intermediate control signals (those valued between `0` and `1`) is set by linear interpolation between this value and that of the **Sonic conductance at maximum flow** parameter.

`Critical pressure ratio` — Ratio of downstream and upstream pressures at which the flow first chokes
`0.3` (default) | positive unitless scalar

Pressure ratio at which flow first begins to choke and the flow velocity reaches its maximum, given by the local speed of sound. The pressure ratio is the fraction of the absolute pressure downstream of the resistive element over the absolute pressure upstream of the same component.

`Opening fraction vector` — Control signal values at which to specify the tabulated parameter data
`0 : 0.2 1` (default) | vector of positive unitless numbers

Vector of control signal values at which to specify the chosen measure of flow capacity (sonic conductance, either of the flow coefficients, or restriction area). The control signal is bounded at `0` and `1`, with each value in this range corresponding to an opening fraction of the resistive element. The greater the value, the greater the opening and (generally) the easier the flow.

The opening fractions must increase monotonically across the vector from left to right. The size of this vector must be the same as that of the chosen measure of flow capacity (**Sonic conductance vector**, **Cv coefficient (USCS) vector**, other).

`Sonic conductance vector` — Sonic conductances corresponding to the values in the opening fraction vector
`[1e-05, .32, .64, .96, 1.28, 1.6] L/s/bar` (default) | vector of positive numbers in units of volume/(time*pressure)

Vector of sonic conductances inside the resistive element, with each conductance corresponding to a value in the **Opening fraction vector** parameter. The sonic conductances must increase monotonically from left to right, with greater opening fractions generally translating into greater sonic conductances. The size of the vector must be the same as that of the **Opening fraction vector**.

`Critical pressure ratio vector` — Critical pressure ratios corresponding to the values in the opening fraction vector
`[0.3, 0.3, 0.3, 0.3, 0.3, 0.3]` (default) | vector of positive unitless numbers

Vector of critical pressure ratios at which the flow first chokes, with each critical pressure ratio corresponding to a value in the **Opening fraction vector** parameter. The critical pressure ratio is the fraction of downstream to upstream pressures at which the flow velocity reaches the local speed of sound. The size of the vector must be the same as that of the **Opening fraction vector**.

`Subsonic index` — Exponent used to more accurately characterize mass flow in the subsonic flow regime
`0.5` (default) | positive unitless scalar

Exponent used to more accurately calculate the mass flow rate in the subsonic flow regime as described in ISO 6358. This parameter is treated as a constant independent of the opening fraction specified by the control signal at port **L**.

`Cv coefficient (USCS) at maximum flow` — Cv coefficient corresponding to a maximally open component
`0.4` (default) | positive scalar in units of ft^3/min

Value of the C_v flow coefficient when the control signal specified at port **L** is `1`. The cross-sectional area available for flow is then at a maximum. During simulation the flow coefficient at intermediate control signals (those valued between `0` and `1`) is set by linear interpolation between this value and that of the **Cv coefficient (USCS) at leakage flow** parameter.

`Cv coefficient (USCS) at leakage flow` — Cv coefficient corresponding to a maximally closed component
`1e-6` (default) | positive scalar in units of ft^3/min

Value of the C_v flow coefficient when the control signal specified at port **L** is `0`. The cross-sectional area available for flow is then at a minimum, with only a negligible leakage flow remaining between the ports. The primary purpose of this parameter is to ensure the numerical robustness of the model during simulation. Its exact value is less important than its being a very small number greater than zero.

During simulation the flow coefficient at intermediate control signals (those valued between `0` and `1`) is set by linear interpolation between this value and that of the **Cv coefficient (USCS) at maximum flow** parameter.

`Cv coefficient (USCS) vector` — Cv coefficients corresponding to the values in the opening fraction vector
`[1e-06, .08, .16, .24, .32, .4]` (default) | vector of positive numbers in units of ft^3/min

Vector of C_v flow coefficients expressed in US customary units of ft^3}/min, with each coefficient corresponding to a value in the **Opening fraction vector** parameter. The flow coefficients must increase monotonically from left to right, with greater opening fractions generally translating into greater flow coefficients. The size of the vector must be the same as that of the **Opening fraction vector**.

`Kv coefficient (SI) at maximum flow` — Kv coefficient corresponding to a maximally open component
`0.3` (default) | positive scalar in units of L/min

Value of the K_v flow coefficient when the control signal specified at port **L** is `1`. The cross-sectional area available for flow is then at a maximum. During simulation the flow coefficient at intermediate control signals (those valued between `0` and `1`) is set by linear

interpolation between this value and that of the **Kv coefficient (SI) at leakage flow** parameter.

`Kv coefficient (SI) at leakage flow` — Kv coefficient corresponding to a maximally closed component
`1e-6` (default) | positive scalar in units of L/min

Value of the K_v flow coefficient when the control signal specified at port **L** is `0`. The cross-sectional area available for flow is then at a minimum, with only a neglible leakage flow remaining between the ports. The primary purpose of this parameter is to ensure the numerical robustness of the model during simulation. Its exact value is less important than its being a very small number greater than zero.

During simulation the flow coefficient at intermediate control signals (those valued between `0` and `1`) is set by linear interpolation between this value and that of the **Kv coefficient (SI) at maximum flow** parameter.

`Kv coefficient (USCS) vector` — Kv coefficients corresponding to the values in the opening fraction vector
`1e-06, .06, .12, .18, .24, .3]` (default) | vector of positive numbers in units of L/min

Vector of K_v flow coefficients expressed in SI units of L/min, with each coefficient corresponding to a value in the **Opening fraction vector** parameter. The flow coefficients must increase monotonically from left to right, with greater opening fractions generally translating into greater flow coefficients. The size of the vector must be the same as that of the **Opening fraction vector**.

`Kv coefficient (SI)` — Flow coefficient expressed in SI units
`0.3` (default) | positive unitless scalar

Flow coefficient expressed in SI units of L/min. This parameter measures the ease with which the gas traverses the resistive element when driven by a pressure differential. See the block description for the correspondence between these parameters.

`Restriction area at maximum flow` — Flow area corresponding to a maximally open component
`1e-5 m^2` (default) | positive scalar in units of area

Value of the flow area at the point of shortest aperture when the control signal specified at port **L** is `1`. The cross-sectional area available for flow is then at a maximum. During simulation the flow area at intermediate control signals (those valued between `0` and `1`) is

set by linear interpolation between this value and that of the **Restriction area at leakage flow** parameter.

`Restriction area at leakage flow` — Flow area corresponding to a maximally closed component
`1e-10 m^2` (default) | positive scalar in units of area

Value of the flow area at the point of shortest aperture when the control signal specified at port **L** is `0`. The cross-sectional area available for flow is then at a minimum, with only a negligible leakage flow remaining between the ports. The primary purpose of this parameter is to ensure the numerical robustness of the model during simulation. Its exact value is less important than its being a very small number greater than zero.

During simulation the flow area at intermediate control signals (those valued between `0` and `1`) is set by linear interpolation between this value and that of the **Restriction area at maximum flow** parameter.

`Restriction area vector` — Flow areas corresponding to the values in the opening fraction vector
`[1e-10, 2e-06, 4e-06, 6e-06, 8e-06, 1e-05] m^2` (default) | vector of positive numbers in units of area

Vector of flow areas as measured at the point of shortest aperture, with each flow area corresponding to a value in the **Opening fraction vector** parameter. The flow areas must increase monotonically from left to right, with greater opening fractions generally translating into greater flow areas. The size of the vector must be the same as that of the **Opening fraction vector**.

`Cross-sectional area at ports A and B` — Flow area at the gas ports
`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the resistive element connects.

`Laminar flow pressure ratio` — Pressure ratio at which flow transitions between laminar and turbulent regimes
`0.999` (default) | positive unitless scalar

Pressure ratio at which flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the resistive

element over the absolute pressure upstream of the same component. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Temperature established in the measurement of sonic conductance
`293.15 K` (default) | positive scalar in units of temperature

Absolute temperature established in the measurement of sonic conductance as defined in ISO 8778.

`Reference density` — Density established in the measurement of sonic conductance
`1.185 kg/m^3` (default) | positive scalar in units of mass/volume

Mass density at the standard conditions established in the measurement of sonic conductance as defined in ISO 8778.

References

[1] P. Beater, *Pneumatic Drives*, Springer-Verlag Berlin Heidelberg, 2007.

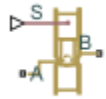
See Also

Orifice ISO 6358 (G)

Introduced in R2018a

Orifice with Variable Area Round Holes

Hydraulic variable orifice shaped as set of round holes drilled in sleeve



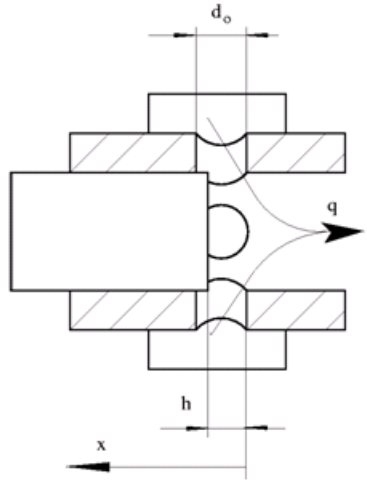
Library

Orifices

Description

The block models a variable orifice created by a cylindrical spool and a set of round holes drilled in the sleeve. All the holes are of the same diameter, evenly spread along the sleeve perimeter, and their center lines are located in the same plane. The flow rate through the orifice is proportional to the orifice opening and to the pressure differential across the orifice. The following schematic shows the cross section of an orifice with variable round holes, where

q	Flow rate
h	Orifice opening
x	Spool displacement from initial position
d_0	Orifice hole diameter



Orifice Opening Area

The aggregate opening area provided by the round holes is computed during simulation from the *orifice opening*, a measure of the instantaneous spool position:

$$h = \begin{cases} h_{\text{Min}}, & \text{if } (x_0 + \delta x) \le h_{\text{Min}} \\ x_0 + \delta x, & \text{else} \end{cases}$$

where:

- h is the orifice opening. The subscript `Min` refers to the minimum value corresponding to a fully closed orifice. This value is calculated during simulation from block parameters.
- x is the spool displacement specified through physical signal port **S**. The subscript `0` denotes the opening offset specified in the **Initial opening** block parameter.
- δ is the orifice opening orientation: `+1` if positive, `-1` if negative. This value is specified in the **Orifice orientation** block parameter.

The minimum orifice opening h_{Min} is calculated approximately using the expression:

$$h_{\mathrm{Min}} \approx \frac{d_{\mathrm{o}}}{2}\left[1-\cos\left(\frac{\pi D_{\mathrm{h,Min}}}{2n_{\mathrm{o}}d_{\mathrm{o}}}\right)\right]$$

where:

- d_{o} is the diameter of an individual hole. This value is obtained from the **Diameter of round holes** block parameter. The holes are assumed to be identical in size.
- n_{o} is the number of round holes comprising the orifice. This value is obtained from the **Number of round holes** block parameter.
- $D_{\mathrm{h,Min}}$ is the minimum value of the hydraulic diameter of the orifice as a whole. This value is calculated from the minimum opening area specified in the **Leakage area** block parameter:

$$D_{\mathrm{h,Min}} = \frac{\pi d_{\mathrm{o}}^2}{4}.$$

The opening area of the orifice (the sum over the various round holes) is calculated from the conditional expression:

$$A(h) = \begin{cases} A_{\mathrm{Leak}} & \text{for } h \le 0 \\ \left[\frac{1}{8}n_{\mathrm{o}}\pi d_{\mathrm{o}}^2\left[2\arccos\left(1-\frac{2h}{d_{\mathrm{o}}}\right)-\sin\left(2\arccos\left(1-\frac{2h}{d_{\mathrm{o}}}\right)\right)\right]\right]+A_{\mathrm{Leak}} & \text{for } 0 < h < d_{\mathrm{o}} \\ A_{Max}+A_{\mathrm{Leak}} & \text{for } h \ge d_{\mathrm{o}} \end{cases},$$

where:

- A_{Leak} is the opening area in the fully closed position, when only leakage remains. This value is obtained from the **Leakage area** block parameter.
- A_{Max} is the opening area in the fully open position, when the holes are open to full capacity. This value is calculated from the number and diameter of the holes:

$$A_{\mathrm{Max}} = n_{\mathrm{o}}\pi\frac{d_{\mathrm{o}}^2}{4}$$

Orifice Flow Rate

The net flow rate through the orifice is calculated as:

$$q = C_D \cdot A(h)\sqrt{\frac{2}{\rho}}\frac{\Delta p}{\left(\Delta p^2 + p_{\mathrm{Cr}}^2\right)^{1/4}},$$

where:

- q is the volumetric flow rate through the orifice (the sum of the the flow rates through the various holes).
- C_D is the discharge coefficient of the orifice. This value is obtained from the **Flow discharge coefficient** block parameter.
- A is the aggregate opening area of the orifice (calculated as a function of the orifice opening h).
- Δp is the pressure differential from port **A** to port **B**:

  $$\Delta p = p_\mathrm{A} - p_\mathrm{B},$$

  where p is the gauge pressure through the port indicated in the subscript.
- p_Cr is the calculated value of the critical pressure (from port **A** to port **B**) at which the flow is considered to shift between laminar and turbulent regimes.

The calculation of the critical pressure differential varies with the setting of the **Laminar transition specification** block parameter:

- `By pressure ratio` — Obtain the critical pressure differential from the critical pressure ratio:

  $$p_\mathrm{Cr} = \left(p_\mathrm{Avg} + p_\mathrm{Atm}\right)(1-\beta),$$

  where β is the critical pressure ratio, obtained from the **Laminar flow pressure ratio** block parameter, defined as the ratio of the outlet pressure to the inlet pressure at the transition point between laminar and turbulent regimes. The subscript `Atm` denotes the atmospheric value (`0.101325 MPa` at standard reference conditions) and the subscript `Avg` the average of the gauge pressures at the ports:

$$p_{\mathrm{Avg}} = \frac{p_{\mathrm{A}} + p_{\mathrm{B}}}{2},$$

- `By Reynolds number` — Obtain the critical pressure differential from the critical Reynolds number:

$$p_{\mathrm{Cr}} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{\mathrm{Cr}} \cdot \nu}{C_D \cdot D_H}\right)^2,$$

where Re_{Cr} is the critical Reynolds number, at which the flow can be said to transition between laminar and turbulent regimes, ν is the average of the kinematic viscosities at the ports, and D_{h} is the hydraulic diameter of the orifice:

$$D_{\mathrm{h}} = \begin{cases} D_{\mathrm{h,Min}}, & \text{if } h \le h_{\mathrm{Min}} \\ d_{\mathrm{o}} + D_{\mathrm{h,Min}}, & \text{if } h > d_{\mathrm{o}} \\ \dfrac{4A}{P}, & \text{else} \end{cases},$$

with P as the sum of the perimeters of the holes:

$$P = n_{\mathrm{o}} d_{\mathrm{o}} \left[2\mathrm{acos}\left(1 - \frac{2h}{d_{\mathrm{o}}}\right) + 2\mathrm{asin}\left(\mathrm{acos}\left[1 - \frac{2h}{d_{\mathrm{o}}}\right]\right)\right]$$

Basic Assumptions and Limitations

- Fluid inertia is ignored.

Parameters

Diameter of round holes

Diameter of the orifice holes. The default value is `5e-3` m.

Number of round holes

Number of holes. The default value is `6`.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.6`.

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is `0`.

Orifice orientation

The parameter is introduced to specify the effect of the orifice control member motion on the valve opening. The parameter can be set to one of two options: `Opens in positive direction` or `Opens in negative direction`. The value `Opens in positive direction` specifies an orifice whose control member opens the valve when it is shifted in the globally assigned positive direction. The parameter is extremely useful for building a multi-orifice valve with all the orifices being controlled by the same spool. The default value is `Opens in positive direction`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `10`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-15 m^2`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Orifice orientation**

All other block parameters are available for modification.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the orifice inlet.

`B`

Hydraulic conserving port associated with the orifice outlet.

`S`

Physical signal port to control spool displacement.

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

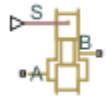
See Also

Annular Orifice | Constant Area Hydraulic Orifice | Fixed Orifice | Orifice with Variable Area Slot | Variable Area Hydraulic Orifice | Variable Orifice

Introduced in R2006a

Orifice with Variable Area Slot

Hydraulic variable orifice shaped as rectangular slot



Library

Orifices

Description

The block models a variable orifice created by a cylindrical sharp-edged spool and a rectangular slot in a sleeve. The flow rate through the orifice is proportional to the orifice opening and to the pressure differential across the orifice. The flow rate is determined according to the following equations:

$$q = C_D \cdot A(h)\sqrt{\frac{2}{\rho}}\frac{\Delta p}{\left(\Delta p^2 + p_{\mathrm{Cr}}^2\right)^{1/4}},$$

$$\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}},$$

$$h = x_0 + x \bullet or$$

$$A(h) = \begin{cases} A_{leak} & \text{if } b \cdot h < A_{leak} \\ b \cdot h & \text{otherwise} \end{cases}$$

where

q	Flow rate

p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
b	Width of the orifice slot
x_0	Initial opening
x	Spool displacement from initial position
h	Orifice opening
or	Orifice orientation indicator. The variable assumes +1 value if a spool displacement in the globally assigned positive direction opens the orifice, and –1 if positive motion decreases the opening.
ρ	Fluid density
A_{leak}	Closed orifice leakage area
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

  $$p_{avg} = (p_A + p_B)/2$$

  where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as

$\Delta p = p_A - p_B$, . Positive signal at the physical signal port `S` opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.

Parameters

Orifice width

The width of the rectangular slot. The default value is `1e-2` m.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is `0`.

Orifice orientation

The parameter is introduced to specify the effect of the orifice control member motion on the valve opening. The parameter can be set to one of two options: `Opens in positive direction` or `Opens in negative direction`. The value `Opens in positive direction` specifies an orifice whose control member opens the valve when it is shifted in the globally assigned positive direction. The parameter is extremely useful for building a multi-orifice valve with all the orifices being controlled by the same spool. The default value is `Opens in positive direction`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12 m^2`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Orifice orientation**

All other block parameters are available for modification.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port to control spool displacement.

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

See Also

Annular Orifice | Constant Area Hydraulic Orifice | Fixed Orifice | Orifice with Variable Area Round Holes | Variable Area Hydraulic Orifice | Variable Orifice

Introduced in R2006a

Partially Filled Vertical Pipe LP

Partially filled vertical pipe connecting two tanks

Library

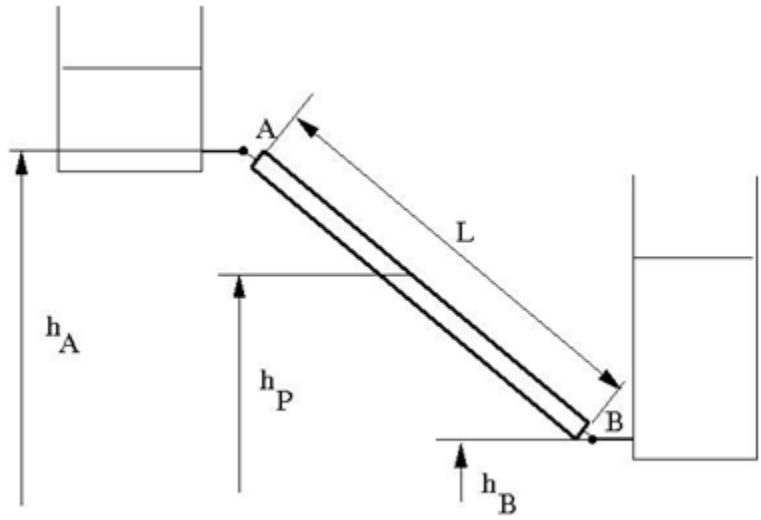
Low-Pressure Blocks

Description

The Partially Filled Vertical Pipe LP block models a vertical pipe connecting two tanks. The block takes into account the possibility that the top tank gets emptied, and therefore the top port of the pipe can be exposed. The fluid level in the pipe then settles at some intermediate position, depending on pressures at both ports.

The following schematic diagram shows two tanks connected by a partially filled vertical pipe, where:

- h_A and h_B are the elevations of ports A and B, respectively
- h_P is the fluid level in the pipe
- L is the length of the pipe



The tank is assumed to be empty when fluid volume becomes less than a specified value. If the upstream tank is emptied, the flow rate through port A is assumed to be 0, and the tank pressurization pressure propagates through port A to the pipe. When pressure increases at the bottom port B, the fluid flows upstream. After the pipe is completely filled, the fluid can start filling the top tank.

You must connect port A to a component that provides data on fluid volume (such as the Reservoir block or any of the tank blocks). Port A elevation must be greater than port B elevation.

You can connect port B to any hydraulic component, with one restriction: the component must keep the port submerged at all times.

The pipe hydraulic resistance is proportional to the length of fluid in the pipe (wet length). The shorter the wet length, the lower the resistance. To avoid zero resistance as wet length approaches 0, the volume of fluid in the pipe is limited to the same minimum value as the tank. After the minimum volume is reached, the flow rate through port B is assumed to be 0.

It is a good practice to connect port B either to another tank or to a pressure source. When you connect it to a pressure source, it is good practice to insert some hydraulic

resistance (fixed orifice, valve, and so on) between the pipe and the source, to avoid zero resistance.

The block models pipes with circular and noncircular cross sections. The block accounts for friction losses, variable fluid level in the pipe, and resulting variable heads at the ports. The block is based on the assumption of steady-state fluid momentum conditions. For additional information, see “Basic Assumptions and Limitations” on page 1-841. To account for local resistances (such as bends, fittings, inlet and outlet losses, and so on), convert the resistances into their equivalent lengths, sum up all the resistances to obtain their aggregate length, and then add this length to the pipe geometrical length. The ratio between the added and the current wet length of the pipe is maintained constant as the pipe is emptied or filled with fluid.

Flow rates through the pipe ports are computed individually because they are different when the top port is exposed. The flow rates are proportional to the flow-regime-dependent friction factor and the pressure differential. The friction factor in turbulent regime is determined with the Haaland approximation [1 on page 1-844]:

$$f = \frac{1}{\left(-1.8\log_{10}\left(\frac{6.9}{Re} + \left(\frac{k/D_H}{3.7}\right)^{1.11}\right)\right)^2}$$

$$\mathrm{Re} = \frac{q \cdot D_H}{A \cdot \nu}$$

where

f	Friction factor
k	Height of the roughness on the pipe internal surface
Re	Reynolds number
q	Flow rate
D_H	Pipe hydraulic diameter
A	Pipe cross-sectional area
ν	Fluid kinematic viscosity

At laminar regime, the friction factor is determined as

$$f = \frac{s}{\mathrm{Re}}$$

where s is the geometrical shape factor, or Poiseuille number [1 on page 1-844], defined as a function of the Fanning factor for various noncircular shapes: rectangle, concentric annulus, ellipse, and circular sector.

The friction factor during the transition from laminar to turbulent regimes is determined with the linear interpolation between extreme points of the regimes. As a result of these assumptions, the pipe is simulated according to the following equations:

$$q_A = \begin{cases} 0 & \text{for } p >= 0,\ V < V_{\min} \\ A\sqrt{\frac{1}{f}\frac{D_H}{L_{ef}}}\sqrt{\frac{2}{\rho}\cdot p} & \text{for } p >= 0,\ V >= V_{\min},\ Re >= Re_{cr} \\ 2\cdot A\cdot\frac{D_H^2}{s\cdot\nu\cdot\rho\cdot L_{ef}}\cdot p & \text{for } p >= 0,\ V >= V_{\min},\ Re < Re_{cr} \\ 0 & \text{for } p < 0,\ L_F <= L \\ 2\cdot A\cdot\frac{D_H^2}{s\cdot\nu\cdot\rho\cdot L_{ef}}\cdot p & \text{for } p < 0,\ L_F > L,\ Re < Re_{cr} \\ A\sqrt{\frac{1}{f}\frac{D_H}{L_{ef}}}\sqrt{\frac{2}{\rho}\cdot |p|} & \text{for } p < 0,\ L_F > L,\ Re >= Re_{cr} \end{cases}$$

$$q_B = \begin{cases} 0 & \text{for } p >= 0,\ V_F < V_{\min} \\ A\sqrt{\frac{1}{f}\frac{D_H}{L_{ef}}}\sqrt{\frac{2}{\rho}\cdot |p|}\cdot sign(p) & \text{for } p >= 0,\ V_F >= V_{\min},\ Re >= Re_{cr} \\ 2\cdot A\cdot\frac{D_H^2}{s\cdot\nu\cdot\rho\cdot L_{ef}}\cdot p & \text{for } p >= 0,\ V_F >= V_{\min},\ Re < Re_{cr} \end{cases}$$

$$q_A - q_B = \frac{dV_p}{dt}$$

$$L_F = \frac{V_p}{A}$$

$$L_{ef} = L_F\left(1 + \frac{L_{ad}}{L}\right)$$

$$h_p = L_F\frac{h_A - h_B}{L}$$

$$p = p_A - p_B + \rho \cdot g \cdot h_p$$

where

q_A, q_B	Flow rates through port A and B, respectively
p_A, p_B	Pressures at port A and B, respectively
h_A, h_B	Port A and port B elevations, respectively
L	Geometrical pipe length
L_{ad}	Aggregate equivalent length of local resistances
L_F	Pipe wet length
L_{ef}	Effective pipe length
ρ	Fluid density
p	Pressure differential across the pipe
V	Fluid volume in the upstream tank
V_p	Volume of fluid in the pipe
V_{min}	Minimum volume of fluid in the tank, or in the pipe. If V becomes less than V_{min}, then the upstream tank is considered empty, the flow rate through port A is assumed to be 0, and the tank pressurization pressure propagates through port A to the pipe. If V_p becomes less than V_{min}, then the pipe is considered empty and the flow rate through port B is assumed to be 0.
h_p	Fluid elevation in the pipe
Re_{cr}	Critical Reynolds number
g	Gravity acceleration

Flow rate A is considered positive when fluid flows into the pipe. Flow rate B is considered positive when fluid flows out of the pipe.

Basic Assumptions and Limitations

- The block is based on the assumption of steady-state fluid momentum conditions.
- Fluid inertia and fluid compressibility are not taken into account.
- Flow is assumed to be fully developed along the pipe length, therefore the end effects are not considered.
- To account for local resistances in the pipe, you must convert the resistances into their equivalent lengths, sum them up to obtain their aggregate length, and then add this aggregate length to the pipe geometrical length.
- The elevation of port A must be greater than that of port B.
- Port A must be connected to the hydraulic port of the top tank. The fluid volume in the top tank must be input into the Partially Filled Vertical Pipe LP block through the physical signal port V. If you use one of the standard Simscape Fluids reservoir or tank blocks, connect its physical output port directly to the input port V. If you use a custom tank block, it must have a port that exports fluid volume in the tank as a physical signal, and you must connect this output port to the input port V.
- Port B must not be exposed.
- The friction factor at turbulent regime is computed with the Haaland approximation.
- The friction factor at laminar regime is determined as the Poiseuille number divided by the Reynolds number.
- The friction factor during transition from laminar to turbulent regime is computed with the linear interpolation.

Parameters

- “Basic Parameters Tab” on page 1-842
- “Vertical Position Tab” on page 1-843

Basic Parameters Tab

Pipe type

The type of pipe cross section: `Circular` or `Noncircular`. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is `Circular`.

Pipe internal diameter

Pipe internal diameter. The parameter is available if **Pipe type** is set to `Circular`. The default value is `0.01` m.

Noncircular pipe cross-sectional area

Pipe cross-sectional area. The parameter is available if **Pipe type** is set to `Noncircular`. The default value is `0.08` m^2.

Noncircular pipe hydraulic diameter

Hydraulic diameter of the pipe cross section. The parameter is available if **Pipe type** is set to `Noncircular`. The default value is `0.1` m.

Geometrical shape factor

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1 on page 1-844]. The default value is `64`, which corresponds to a pipe with a circular cross section.

Pipe length

Pipe geometrical length. The default value is `100` m.

Aggregate equivalent length of local resistances

Represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is `50` m.

Tank minimum volume

Minimum volume of fluid in the upstream tank, or in the pipe, corresponding to V_{min} in the block equations. The tank or pipe is considered empty if its fluid volume becomes less than V_{min}. The default value is `1e-4` m^3.

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer catalogs. The default value is `5e-5` m.

Vertical Position Tab

Port A elevation wrt reference plane

The vertical position of the pipe port A with respect to the reference plane. The default value is `50` m.

Port B elevation wrt reference plane

The vertical position of the pipe port B with respect to the reference plane. The default value is `0`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Pipe type**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Pipe type** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

V

Physical signal input port that provides data on fluid volume in the upstream tank.

L

Physical signal output port that exports the pipe fluid level.

References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

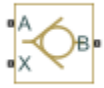
See Also

Hydraulic Pipe LP with Variable Elevation | Resistive Pipe LP with Variable Elevation

Introduced in R2013a

Pilot-Operated Check Valve

Hydraulic check valve that allows flow in one direction, but can be disabled by pilot pressure

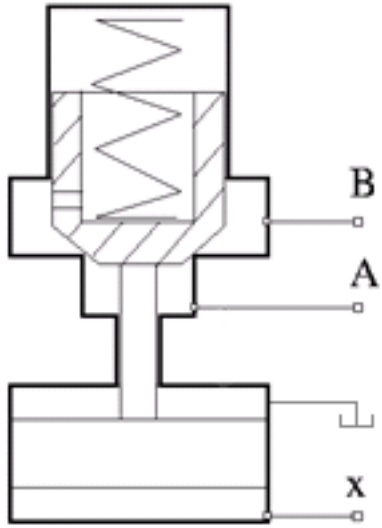


Library

Directional Valves

Description

The Pilot-Operated Check Valve block represents a hydraulic pilot-operated check valve as a data-sheet-based model. The purpose of the check valve is to permit flow in one direction and block it in the opposite direction, as shown in the following figure.



Unlike a conventional check valve, the pilot-operated check valve can be opened by inlet pressure p_A, pilot pressure p_X, or both. The force acting on the poppet is based on the expression

$$F = p_A \bullet A_A + p_X \bullet A_X - p_B \bullet A_B$$

where

p_A, p_B	Gauge pressures at the valve terminals
p_X	Pressure—gauge or differential—at the pilot terminal
A_A	Area of the spool in the A chamber
A_B	Area of the spool in the B chamber
A_X	Area of the pilot chamber

The force equation is commonly expressed in the slightly modified form

$$p_e = p_A + p_X \bullet k_p - p_B$$

where $k_p = A_X / A_A$ is usually referred to as pilot ratio and p_e is the effective pressure differential across the control member. The valve remains closed while this pressure differential across the valve is lower than the valve cracking pressure. When cracking pressure is reached, the valve control member is forced off its seat, thus creating a passage between the inlet and outlet. If the flow rate is high enough and pressure continues to rise, the area is further increased until the control member reaches its maximum. At this moment, the valve passage area is at its maximum. The valve maximum area and the cracking and maximum pressures are generally provided in the catalogs and are the three key parameters of the block.

The pilot pressure can be a differential value relative to the inlet (port **A**) or a gauge value (relative to the environment). You can select an appropriate setting—**Pressure differential (pX - pA)** or **Pressure at port X**—using the **Pressure control specification** dropdown list. If **Pressure at port X** is selected:

$$p_X = p_{X,Abs} - p_{Atm},$$

where the subscript `Atm` denotes atmospheric pressure. The subscript `X,Abs` denotes the absolute value at the pilot port. If **Pressure differential (pX - pA)** is selected:

$$p_X = p_{X,Abs} - p_{A,Abs}$$

where the subscript `A,Abs` similarly denotes the absolute value at the inlet of the valve (port **A**). The pilot pressure differential is constrained to be greater than or equal to zero —if its calculated value should be negative, zero is assumed in the control pressure calculation.

In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Therefore, the parameter value must be greater than zero.

By default, the block does not include valve opening dynamics, and the valve sets its opening area directly as a function of pressure:

$$A = A(p)$$

Adding valve opening dynamics provides continuous behavior that is more physically realistic, and is particularly helpful in situations with rapid valve opening and closing. The pressure-dependent orifice passage area *A(p)* in the block equations then becomes the steady-state area, and the instantaneous orifice passage area in the flow equation is determined as follows:

$$A(t=0) = A_{init}$$

$$\frac{dA}{dt} = \frac{A(p) - A}{\tau}$$

In either case, the flow rate through the valve is determined according to the following equations:

$$q = C_D \cdot A\sqrt{\frac{2}{\rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

$$p_e = p_A + p_X \bullet k_p - p_B$$

$$A(p) = \begin{cases} A_{leak} & \text{for } p_e <= p_{crack} \\ A_{leak} + k \bullet \left(p_e - p_{crack}\right) & \text{for } p_{crack} < p_e < p_{\max} \\ A_{\max} & \text{for } p_e >= p_{\max} \end{cases}$$

$$k = \frac{A_{\max} - A_{leak}}{p_{\max} - p_{crack}}$$

$$\Delta p = p_A - p_B,$$

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr} \cdot \nu}{C_D \cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

q	Flow rate through the valve
p	Pressure differential across the valve
p_e	Equivalent pressure differential across the control member
p_A, p_B	Gauge pressures at the valve terminals
p_X	Gauge pressure at the pilot terminal
k_p	Pilot ratio, $k_p = A_X / A_A$
k	Valve gain coefficient
C_D	Flow discharge coefficient
A	Instantaneous orifice passage area
$A(p)$	Pressure-dependent orifice passage area
A_{init}	Initial open area of the valve
A_{max}	Fully open valve passage area
A_{leak}	Closed valve leakage area
p_{crack}	Valve cracking pressure
p_{max}	Pressure needed to fully open the valve
p_{cr}	Minimum pressure for turbulent flow
Re_{cr}	Critical Reynolds number
D_H	Instantaneous orifice hydraulic diameter
ρ	Fluid density
ν	Fluid kinematic viscosity
τ	Time constant for the first order response of the valve opening
t	Time

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as

$\Delta p = p_A - p_B$, .

Basic Assumptions and Limitations

- Valve opening is linearly proportional to the effective pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.
- No flow consumption is associated with the pilot chamber.

Parameters

Pressure control specification

Choice of pressure measurement to use as pilot control signal. The block uses the chosen measurement to calculate the effective pressure differential across the valve. (See the block description for the calculations.)

In the default setting (`Pressure differential (pX - pA)`), the effective pressure differential is a function of the pressure drop from the pilot port (**X**) to the inlet (**A**). In the alternate setting (**Pressure at port X**), it is a function of the gauge pressure at the inlet.

Cracking pressure

Pressure level at which the orifice of the valve starts to open. The default value is `3e4` Pa.

Maximum opening pressure

Pressure differential across the valve needed to fully open the valve. Its value must be higher than the cracking pressure. The default value is `1.2e5` Pa.

Pilot ratio

Ratio between effective area in the pilot chamber to the effective area in the inlet chamber. The default value is `5`.

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Opening dynamics

Select one of the following options:

- `Do not include valve opening dynamics` — The valve sets its orifice passage area directly as a function of pressure. If the area changes instantaneously, so does the flow equation. This is the default.
- `Include valve opening dynamics` — Provide continuous behavior that is more physically realistic, by adding a first-order lag during valve opening and closing. Use this option in hydraulic simulations with the local solver for real-time simulation. This option is also helpful if you are interested in valve opening dynamics in variable step simulations.

Opening time constant

The time constant for the first order response of the valve opening. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `0.1` s.

Initial area

The initial opening area of the valve. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `1e-12` m^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Opening dynamics**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

X

Hydraulic conserving port associated with the valve pilot terminal.

See Also

Check Valve

Introduced in R2006a

Pilot-Operated Check Valve (TL)

Check valve with control port to enable flow in reverse direction

Library: Simscape / Fluids / Thermal Liquid / Valves & Orifices / Directional Control Valves

Description

The Pilot-Operated Check Valve (TL) block models a check valve with an override mechanism to allow for backflow when activated. (A check valve in turn is an orifice with a unidirectional opening mechanism installed to prevent just that backflow.)

The override mechanism adds a third port—the *pilot*—to the valve. During normal operation, the pilot port is inactive and the valve behaves as any other check valve. Its orifice is then open only when the pressure gradient across it drops from inlet to outlet. Backflow, which requires the reverse pressure gradient, cannot occur. This mode protects components upstream of the valve against pressure surges, temperature spikes, and (in real systems) chemical contamination arising from points downstream.

When backflow is desired, the pilot port is pressurized and the control element of the valve—often a ball or piston—is forced off its seat. The valve is then open to flow in both directions, with a reverse pressure drop (aimed from outlet to inlet) sufficing to drive the flow upstream. (The seat, which lies in the path of the flow, determines if the valve is open. When it is covered—by a ball, piston, or other control element—the flow is cut off and the valve is closed.)

The valve opens by degrees, beginning at its cracking pressure, and continuing to the end of its pressure regulation range. The cracking pressure gives the initial resistance, due to friction or spring forces, that the valve must overcome to open by a sliver (or to *crack* open). Below this threshold, the valve is closed and only leakage flow can pass. Past the end of the pressure regulation range, the valve is fully open and the flow at a maximum (determined by the instantaneous pressure conditions).

The cracking pressure assumes an important role in check valves installed upside down. There, the weight of the opening element—such as a ball or piston—and the elevation head of the fluid can act to open the valve. (The elevation head can arise in a model from

a pipe upstream of the inlet when it is vertical or given a slant.) A sufficient cracking pressure keeps the valve from opening inadvertently even if placed at a disadvantageous angle.

Control Pressure

The opening of the valve depends both on the pilot pressure and on the pressure drop from inlet to outlet:

$$p_{\mathrm{Ctl}} = k_{\mathrm{X}} p_{\mathrm{X}} + p_{\mathrm{A}} - p_{\mathrm{B}},$$

where p is gauge pressure and k is the pilot ratio—the proportion of the pilot opening area (S_{X}) to the valve opening area (S_{R}). The subscript `X` denotes a pilot value and the subscripts `A` and `B` the inlet and outlet values, respectively. The port pressures are variables determined (against absolute zero) during simulation.

The pilot pressure can be a differential value relative to the inlet (port **A**) or a gauge value (relative to the environment). You can select an appropriate setting—**Pressure differential (pX - pA)** or **Pressure at port X**—using the **Pressure control specification** dropdown list. If **Pressure at port X** is selected:

$$p_{\mathrm{X}} = p_{\mathrm{X,Abs}} - p_{\mathrm{Atm}},$$

where the subscript `Atm` denotes the atmospheric value (obtained from the Thermal Liquid Settings (TL) or Thermal Liquid Properties (TL) block of the model). The subscript `X,Abs` denotes the absolute value at the pilot port. If **Pressure differential (pX - pA)** is selected:

$$p_{\mathrm{X}} = p_{\mathrm{X,Abs}} - p_{\mathrm{A,Abs}}$$

where the subscript `A,Abs` similarly denotes the absolute value at the inlet of the valve (port **A**). The pilot pressure differential is constrained to be greater than or equal to zero —if its calculated value should be negative, zero is assumed in the control pressure calculation.

Control Pressure Overshoot

The degree to which the control pressure exceeds the cracking pressure determines how much the valve will open. The pressure overshoot is expressed here as a fraction of the (width of the) pressure regulation range:

$$\hat{p} = \frac{p_{\mathrm{Ctl}} - P_{\mathrm{Crk}}}{P_{\mathrm{Max}} - P_{\mathrm{Crk}}}.$$

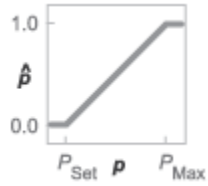
The control pressure (p_{Ctl}), cracking pressure (p_{Set}), and maximum opening pressure (P_{Max}) correspond to the control pressure specification chosen (`Pressure differential` or `Pressure at port A`).

The fraction—technically, the overshoot *normalized*—is valued at `0` in the fully closed valve and `1` in the fully open valve. If the calculation should return a value outside of these bounds, the nearest of the two is used instead. (In other words, the fraction *saturates* at `0` and `1`.)

The normalized control pressure, p, spans three pressure regions. Below the cracking pressure of the valve, its value is a constant zero. Above the maximum pressure of the same, it is `1`. In between, it varies, as a linear function of the (effective) control pressure, p_{Ctl}.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish the simulation run, perhaps excessively so for practical use in real-time simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized temperature overshoot with sharp transitions

The block removes the discontinuities by smoothing them over a specified time scale. The smoothing, which adds a slight distortion to the normalized inlet pressure, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_{\mathrm{L}} = 3\bar{p}_{\mathrm{L}}^{2} - 2\bar{p}_{\mathrm{L}}^{3}$$

and

$$\lambda_{\mathrm{R}} = 3\bar{p}_{\mathrm{R}}^{2} - 2\bar{p}_{\mathrm{R}}^{3},$$

where

$$\bar{p}_{\mathrm{L}} = \frac{\hat{p}}{\Delta p^{*}}$$

and

$$\bar{p}_{\mathrm{R}} = \frac{\hat{p} - \left(1 - \Delta p^{*}\right)}{\Delta p^{*}}.$$

In the equations:

- λ_{L} is the smoothing expression for the transition from the maximally closed position.
- λ_{R} is the smoothing expression for the transition from the fully open position.
- Δp^{*} is the (unitless) characteristic width of the pressure smoothing region:

$$\Delta p^* = f^* \frac{1}{2},$$

where f^* is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

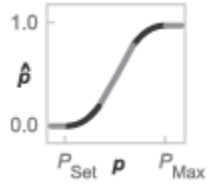
When the smoothing factor is `0`, the normalized inlet pressure stays in its original form —no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the pressure regulation range (with the normalized inlet pressure taking the shape of an *S*-curve).

At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the pressure regulation range on each side (for a total smooth region of half the regulation range).

The smoothing adds two new regions to the normalized pressure overshoot—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$\hat{p}^* = \begin{cases} 0, & \hat{p} \le 0 \\ \hat{p}\lambda_{\mathrm{L}}, & \hat{p} < \Delta P^* \\ \hat{p}, & \hat{p} \le 1 - \Delta P^* \\ \hat{p}\left(1 - \lambda_{\mathrm{R}}\right) + \lambda_{\mathrm{R}}, & \hat{p} < 1 \\ 1 & \hat{p} \ge 1 \end{cases},$$

where the asterisk denotes a smoothed variable (the normalized control pressure overshoot). The figure shows the effect of smoothing on the sharpness of the transitions.



Opening Area

The valve is assumed to open linearly with the smoothed control pressure overshoot:

$$S = \left(S_{\mathrm{Max}} - S_{\mathrm{Min}}\right)\hat{p} + S_{\mathrm{Min}},$$

where S is sonic conductance and the subscripts `Max` and `Min` denote its values in the fully open and fully closed valve. In terms of the smoothed control pressure overshoot, the opening area becomes:

$$S^{*} = \left(S_{\mathrm{Max}} - S_{\mathrm{Min}}\right)p^{*} + S_{\mathrm{Min}}.$$

(Both expressions are used in the calculation of the pressure drop across the valve.)

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is captured in the block by the discharge coefficient, a measure of the mass flow rate through the valve relative to the theoretical value that it would have in an ideal valve. Expressing the momentum balance in the valve in terms of the pressure drop induced in the flow:

$$p_{\mathrm{A}} - p_{\mathrm{B}} = \frac{\dot{m}_{\mathrm{Avg}}\sqrt{\dot{m}_{\mathrm{Avg}}^{2} + \dot{m}_{\mathrm{Crit}}^{2}}}{2\rho_{\mathrm{Avg}}C_{\mathrm{D}}S^{*2}}\left[1 - \left(\frac{S^{*}}{S}\right)^{2}\right]\xi_{\mathrm{p}},$$

where ρ is density, C_{D} is the discharge coefficient, and ξ_{p} is the pressure drop ratio—a measure of the extent to which the pressure recovery at the outlet contributes to the total pressure drop of the valve. The subscript Avg denotes an average of the values at the thermal liquid ports. The critical mass flow rate, $\dot{m}_{\mathrm{Crit}}$, is calculated from the critical Reynolds number—that at which the flow in the valve is assumed to transition from laminar to turbulent:

$$\dot{m}_{\mathrm{Crit}} = \mathrm{Re}_{\mathrm{Crit}}\mu_{\mathrm{Avg}}\sqrt{\frac{\pi}{4}S},$$

where μ denotes dynamic viscosity. The pressure drop ratio is calculated as:

$$\xi_{\mathrm{p}} = \frac{\sqrt{1-\left(\frac{S^*}{S}\right)^2\left(1-C_{\mathrm{D}}^2\right)} - C_{\mathrm{D}}\frac{S^*}{S}}{\sqrt{1-\left(\frac{S^*}{S}\right)^2\left(1-C_{\mathrm{D}}^2\right)} + C_{\mathrm{D}}\frac{S^*}{S}}.$$

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of fluid can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the fluid and the wall that surrounds it. No work is done on or by the fluid as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Valve entrance
thermal liquid

Opening through which the working fluid must, during normal operation (when the pilot mechanism is disabled), enter the valve. This port can serve as an exit only when the pilot port is pressurized to a sufficient degree.

B — Valve entrance
thermal liquid

Opening through which the working fluid must, during normal operation (when the pilot mechanism is disabled), exit the valve.

X — Pilot port
thermal liquid

Opening by which to actuate (by the application of a sufficient pressure) the pilot mechanism that opens the valve for reverse flow.

Parameters

`Pressure control specification` — Choice of pressure measurement to use as valve control signal
`Pressure differential (pX-pA)` (default) | `Pressure at port X`

Choice of pressure measurement to use as pilot control signal. The block uses the chosen measurement to calculate the control pressure of the valve. (See the block description for the calculations.)

In the default setting (`Pressure differential (pX - pA)`), the control pressure is a function of the pressure drop from the pilot port (**X**) to the inlet (**A**). In the alternate setting (**Pressure at port X**), it is a function of the gauge pressure at the inlet.

`Cracking pressure differential` — Effective control pressure at which the valve first opens
`0.01 MPa` (default) | positive scalar in units of pressure

Effective pressure differential at which the valve begins to open. This differential—the *control pressure* in the block description—is the sum of the pressure drop from inlet to outlet with the product of the pilot pressure and the pilot ratio. (The pilot pressure used depends on the setting of the **Pressure control specification** parameter.)

`Maximum opening pressure differential` — Effective control pressure at which the valve is fully open
`0.02 MPa` (default) | positive scalar in units of pressure

Effective pressure differential at which the valve is fully open. This differential—the *control pressure* in the block description—is the sum of the pressure drop from inlet to outlet with the product of the pilot pressure and the pilot ratio. (The pilot pressure used depends on the setting of the **Pressure control specification** parameter.)

`Pilot ratio` — Ratio of pilot port area to inlet port area
`1` (default) | positive unitless scalar

Ratio of the pilot port area to the inlet port area.

`Maximum opening area` — Opening area in the fully open position due to sealing imperfections
`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position, when the valve is at the upper limit of the pressure regulation range.

`Leakage area` — Opening area in the maximally closed position due to sealing imperfections
`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the fluid network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

`Smoothing factor` — Amount of smoothing to apply to the valve opening area function
`0` (default)

Amount of smoothing to apply to the opening area function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening area function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings), which tend to slow down the rate of simulation.

`Cross-sectional area at ports A and B` — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be the same in size. The flow area specified here should ideally match those of the inlets of adjoining components.

`Characteristic longitudinal length` — Measure of the length of the flow path through the valve

`0.1` (default) | positive scalar in units of length

Average distance traversed by the fluid as it travels from inlet to outlet. This distance is used in the calculation of the internal thermal conduction that occurs between the two ports (as part of the smoothed upwind energy scheme employed in the thermal liquid domain).

`Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates

`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes

12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

See Also

2-Way Directional Valve (TL) | 3-Way Directional Valve (TL) | Check Valve (TL) | Variable Area Orifice (TL)

Introduced in R2018b

Pilot-Operated Check Valve (G)

Check valve with control port to enable flow in reverse direction

Library: Simscape / Fluids / Gas / Valves & Orifices / Directional Control Valves

Description

The Pilot-Operated Check Valve (G) block models a check valve with an override mechanism to allow for backflow when activated. (A check valve in turn is an orifice with a unidirectional opening mechanism installed to prevent just that backflow.)

The override mechanism adds a third port—the *pilot*—to the valve. During normal operation, the pilot port is inactive and the valve behaves as any other check valve. Its orifice is then open only when the pressure gradient across it drops from inlet to outlet. Backflow, which requires the reverse pressure gradient, cannot occur. This mode protects components upstream of the valve against pressure surges, temperature spikes, and (in real systems) chemical contamination arising from points downstream.

When backflow is desired, the pilot port is pressurized and the control element of the valve—often a ball or piston—is forced off its seat. The valve is then open to flow in both directions, with a reverse pressure drop (aimed from outlet to inlet) sufficing to drive the flow upstream. (The seat, which lies in the path of the flow, determines if the valve is open. When it is covered—by a ball, piston, or other control element—the flow is cut off and the valve is closed.)

The valve opens by degrees, beginning at its cracking pressure, and continuing to the end of its pressure regulation range. The cracking pressure gives the initial resistance, due to friction or spring forces, that the valve must overcome to open by a sliver (or to *crack* open). Below this threshold, the valve is closed and only leakage flow can pass. Past the end of the pressure regulation range, the valve is fully open and the flow at a maximum (determined by the instantaneous pressure conditions).

The cracking pressure assumes an important role in check valves installed upside down. There, the weight of the opening element—such as a ball or piston—and the elevation head of the fluid can act to open the valve. (The elevation head can arise in a model from

a pipe upstream of the inlet when it is vertical or given a slant.) A sufficient cracking pressure keeps the valve from opening inadvertently even if placed at a disadvantageous angle.

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Control and Other Pressures

The opening of the valve depends both on the pilot pressure and on the pressure drop from inlet to outlet:

$$p_{\mathrm{Ctl}} = k_{\mathrm{X}} p_{\mathrm{X}} + p_{\mathrm{A}} - p_{\mathrm{B}},$$

where p is gauge pressure and k is the pilot ratio—the proportion of the pilot opening area (S_{X}) to the valve opening area (S_{R}). The subscript `X` denotes a pilot value and the subscripts `A` and `B` the inlet and outlet values, respectively. The port pressures are variables determined (against absolute zero) during simulation.

The pilot pressure can be a differential value relative to the inlet (port **A**) or a gauge value (relative to the environment). You can select an appropriate setting—**Pressure differential (pX - pA)** or **Pressure at port X**—using the **Pressure control specification** dropdown list. If **Pressure at port X** is selected:

$$p_{\mathrm{X}} = p_{\mathrm{X,Abs}} - p_{\mathrm{Atm}},$$

where the subscript `Atm` denotes the atmospheric value (obtained from the Gas Properties (G) block of the model). The subscript `X,Abs` denotes the absolute value at the pilot port. If **Pressure differential (pX - pA)** is selected:

$$p_{\mathrm{X}} = p_{\mathrm{X,Abs}} - p_{\mathrm{A,Abs}}$$

where the subscript `A,Abs` similarly denotes the absolute value at the inlet of the valve (port **A**). The pilot pressure differential is constrained to be greater than or equal to zero —if its calculated value should be negative, zero is assumed in the control pressure calculation.

Control Pressure Overshoot

The degree to which the control pressure exceeds the cracking pressure determines how much the valve will open. The pressure overshoot is expressed here as a fraction of the (width of the) pressure regulation range:

$$\hat{p} = \frac{p_{\mathrm{Ctl}} - P_{\mathrm{Crk}}}{P_{\mathrm{Max}} - P_{\mathrm{Crk}}}.$$

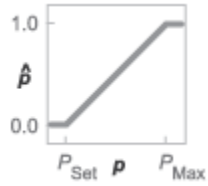
The control pressure (p_{Ctl}), cracking pressure (p_{Set}), and maximum opening pressure (P_{Max}) correspond to the control pressure specification chosen (`Pressure differential` or `Pressure at port A`).

The fraction—technically, the overshoot *normalized*—is valued at `0` in the fully closed valve and `1` in the fully open valve. If the calculation should return a value outside of these bounds, the nearest of the two is used instead. (In other words, the fraction *saturates* at `0` and `1`.)

The normalized control pressure, p, spans three pressure regions. Below the cracking pressure of the valve, its value is a constant zero. Above the maximum pressure of the same, it is `1`. In between, it varies, as a linear function of the (effective) control pressure, p_{Ctl}.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish the simulation run, perhaps excessively so for practical use in real-time simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized pressure overshoot with sharp transitions

The block removes the discontinuities by smoothing them over a specified time scale. The smoothing, which adds a slight distortion to the normalized inlet pressure, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_{\mathrm{L}} = 3\bar{p}_{\mathrm{L}}^{2} - 2\bar{p}_{\mathrm{L}}^{3}$$

and

$$\lambda_{\mathrm{R}} = 3\bar{p}_{\mathrm{R}}^{2} - 2\bar{p}_{\mathrm{R}}^{3},$$

where

$$\bar{p}_{\mathrm{L}} = \frac{\hat{p}}{\Delta p^{*}}$$

and

$$\bar{p}_{\mathrm{R}} = \frac{\hat{p} - \left(1 - \Delta p^{*}\right)}{\Delta p^{*}}.$$

In the equations:

- λ_{L} is the smoothing expression for the transition from the maximally closed position.
- λ_{R} is the smoothing expression for the transition from the fully open position.
- Δp^{*} is the (unitless) characteristic width of the pressure smoothing region:

$$\Delta p^* = f^* \frac{1}{2},$$

where f^* is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

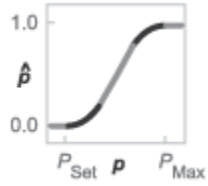
When the smoothing factor is `0`, the normalized inlet pressure stays in its original form —no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the pressure regulation range (with the normalized inlet pressure taking the shape of an *S*-curve).

At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the pressure regulation range on each side (for a total smooth region of half the regulation range).

The smoothing adds two new regions to the normalized pressure overshoot—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$\hat{p}^* = \begin{cases} 0, & \hat{p} \le 0 \\ \hat{p}\lambda_{\mathrm{L}}, & \hat{p} < \Delta P^* \\ \hat{p}, & \hat{p} \le 1 - \Delta P^* \\ \hat{p}\left(1-\lambda_{\mathrm{R}}\right)+\lambda_{\mathrm{R}}, & \hat{p} < 1 \\ 1 & \hat{p} \ge 1 \end{cases},$$

where the asterisk denotes a smoothed variable (the normalized control pressure overshoot). The figure shows the effect of smoothing on the sharpness of the transitions.



Sonic Conductance

As the normalized control pressure varies during simulation, so does the mass flow rate through the valve. The relationship between the two variables, however, is indirect. The mass flow rate is defined in terms of the valve's sonic conductance and it is this quantity that the normalized inlet pressure determines.

Sonic conductance, if you are unfamiliar with it, describes the ease with which a gas will flow when it is choked—when its velocity is at its theoretical maximum (the local speed of sound). Its measurement and calculation are covered in detail in the ISO 6358 standard (on which this block is based).

Only one value is commonly reported in valve data sheets: one taken at steady state in the fully open position. This is the same specified in the **Sonic conductance at maximum flow** parameter when the **Valve parameterization** setting is `Sonic conductance`. For values across the opening range of the valve, this maximum is scaled by the normalized pressure overshoot:

$$C = \left(C_{\mathrm{Max}} - C_{\mathrm{Min}} \right) \hat{p} + C_{\mathrm{Min}},$$

where C is sonic conductance and the subscripts `Max` and `Min` denote its values in the fully open and fully closed valve.

Because sonic conductance may not be available (or the most convenient choice for your model), the block provides several equivalent parameterizations. Use the **Valve parameterization** drop-down list to select the best for the data at hand. The parameterizations are:

- `Restriction area`
- `Sonic conductance`
- `Cv coefficient (USCS)`
- `Kv coefficient (SI)`

The parameterizations differ only in the data that they require of you. Their mass flow rate calculations are still based on sonic conductance. If you select a parameterization other than `Sonic conductance`, then the block converts the alternate data—the (computed) opening area or a (specified) flow coefficient—into an equivalent sonic conductance.

The flow coefficients measure what is, at bottom, the same quantity—the flow rate through the valve at some agreed-upon temperature and pressure differential. They differ only in the standard conditions used in their definition and in the physical units used in their expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Cv coefficient (USCS)`.
- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `1 bar`; it is expressed in metric units of m^3/h. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Kv coefficient (SI)`.

If the valve parameterization is set to `Cv Coefficient (USCS)`, the sonic conductance is computed at the maximally closed and fully open valve positions from the **Cv coefficient (SI) at maximum flow** and **Cv coefficient (SI) at leakage flow** block parameters:

$$C = \left(4 \times 10^{-8} C_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where C_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`. (These are used in the mass flow rate calculations given in the “Momentum Balance” on page 1-872 section.)

If the `Kv coefficient (SI)` parameterization is used instead, the sonic conductance is computed at the same valve positions (maximally closed and fully open) from the **Kv coefficient (USCS) at maximum flow** and **Kv coefficient (USCS) at leakage flow** block parameters:

$$C = \left(4.758 \times 10^{-8} K_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where K_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`.

For the `Restriction area` parameterization, the sonic conductance is computed (at the same valve positions) from the **Maximum opening area**, and **Leakage area** block parameters:

$$C = \left(0.128 \times 4S / \pi\right) \texttt{L/(s bar)},$$

where S is the opening area at maximum or leakage flow. The subsonic index, m, is set to `0.5` while the critical pressure ratio, b_{cr} is computed from the expression:

$$0.41 + 0.272\left[\frac{p\left(S_{\text{Max}} - S_{\text{Leak}}\right) + S_{\text{Leak}}}{S}\right]^{0.25}.$$

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is assumed to reflect entirely in the sonic conductance of the valve (or in the data of the alternate valve parameterizations).

When the flow is choked, the mass flow rate is a function of the sonic conductance of the valve and of the thermodynamic conditions (pressure and temperature) established at the inlet. The function is linear with respect to pressure:

$$\dot{m}_{\text{ch}} = C\rho_0 p_{\text{in}}\sqrt{\frac{T_0}{T_{\text{in}}}},$$

where:

- C is the sonic conductance inside the valve. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- ρ is the gas density, here at standard conditions (subscript `0`), obtained from the **Reference density** block parameter.
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (`in`) or at standard conditions (`0`), the latter obtained from the **Reference temperature** block parameter.

When the flow is subsonic, and therefore no longer choked, the mass flow rate becomes a nonlinear function of pressure—both that at the inlet as well as the reduced value at the

outlet. In the turbulent flow regime (with the outlet pressure contained in the back-pressure ratio of the valve), the mass flow rate expression is:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{p_{\mathrm{r}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m ,$$

where:

- p_{r} is the back-pressure ratio, or that between the outlet pressure (p_{out}) and the inlet pressure (p_{in}):

  $$P_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the flow becomes choked. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

When the flow is laminar (and still subsonic), the mass flow rate expression changes to:

$$\dot{m}_{\mathrm{lam}} = C\rho_0 p_{\mathrm{in}} \left[\frac{1 - p_{\mathrm{r}}}{1 - b_{\mathrm{lam}}} \right] \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{b_{\mathrm{lam}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes (obtained from the **Laminar flow pressure ratio** block parameter). Combining the mass flow rate expressions into a single (piecewise) function, gives:

$$\dot{m} = \begin{cases} \dot{m}_{\mathrm{lam}}, & b_{\mathrm{lam}} \le p_{\mathrm{r}} < 1 \\ \dot{m}_{\mathrm{tur}}, & b_{\mathrm{cr}} \le p_{\mathrm{r}} < p_{\mathrm{lam}} \\ \dot{m}_{\mathrm{ch}}, & p_{\mathrm{r}} < b_{\mathrm{Cr}} \end{cases},$$

with the top row corresponding to subsonic and laminar flow, the middle row to subsonic and turbulent flow, and the bottom row to choked (and therefore sonic) flow.

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of gas can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**. Note that in this block the flow can reach but not exceed sonic speeds.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the gas and the wall that surrounds it. No work is done on or by the gas as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Valve entrance

gas

Opening through which the working fluid must, during normal operation (when the pilot mechanism is disabled), enter the valve. This port can serve as an exit only when the pilot port is pressurized to a sufficient degree.

B — Valve entrance

gas

Opening through which the working fluid must, during normal operation (when the pilot mechanism is disabled), exit the valve.

X — Pilot port
gas

Opening by which to actuate (by the application of a sufficient pressure) the pilot mechanism that opens the valve for reverse flow.

Parameters

`Pressure control specification` — Choice of pressure measurement to use as valve control signal
`Pressure differential (pX-pA)` (default) | `Pressure at port X`

Choice of pressure measurement to use as valve control signal. The block uses this setting to determine when the valve should begin to open. In the default setting (`Pressure differential (pX - pA)`), the opening pressure of the valve is expressed as a pressure drop from the pilot port (**X**) to the inlet (**A**). In the alternative setting (**Pressure at port X**), it is expressed as a gauge inlet pressure (measured against the environment pressure).

`Cracking pressure differential` — Effective control pressure at which the valve first opens
`0.01 MPa` (default) | positive scalar in units of pressure

Effective pressure differential at which the valve begins to open. This differential—the *control pressure* in the block description—is the sum of the pressure drop from inlet to outlet with the product of the pilot pressure and the pilot ratio. (The pilot pressure used depends on the setting of the **Pressure control specification** parameter.)

`Maximum opening pressure differential` — Effective control pressure at which the valve is fully open
`0.02 MPa` (default) | positive scalar in units of pressure

Effective pressure differential at which the valve is fully open. This differential—the *control pressure* in the block description—is the sum of the pressure drop from inlet to outlet with the product of the pilot pressure and the pilot ratio. (The pilot pressure used depends on the setting of the **Pressure control specification** parameter.)

`Pilot ratio` — Ratio of pilot port area to inlet port area
`1` (default) | positive unitless scalar

Ratio of the pilot port area to the inlet port area.

`Valve parameterization` — Method by which to characterize the opening of the valve
`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the **Sonic conductance** parameterization; if a different option is selected, the data specified in converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the block description for more information on the conversion.

This parameter determines which measures of valve opening you must specify—and therefore which of those measures appear as parameters in the block dialog box.

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions
`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value generally reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions
`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum

when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of $\mathtt{ft^3/min}$ (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of $\mathtt{ft^3/min}$ (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Maximum opening area` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position, when the valve is at the upper limit of the pressure regulation range. The block uses this parameter to scale the chosen measure of valve opening—sonic conductance, say, or C_V flow coefficient—throughout the pressure regulation range.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Leakage area` — Opening area in the maximally closed position due to sealing imperfections

`1e-12 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the

valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Cross-sectional area at ports A and B` — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be the same in size. The flow area specified here should ideally match those of the inlets of adjoining components.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance

`293.15 K` (default) | positive scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance

`1.185` (default) | positive scalar in units of density

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Smoothing factor` — Amount of smoothing to apply to the valve opening area function
`0` (default)

Amount of smoothing to apply to the opening area function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening area function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings), which tend to slow down the rate of simulation.

See Also

Check Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

Pipe Bend

Hydraulic resistance in pipe bend

Library

Local Hydraulic Resistances

Description

The Pipe Bend block represents a pipe bend as a local hydraulic resistance. The pressure loss in the bend is assumed to consist of

- Loss in the straight pipe
- Loss due to curvature

The loss in a straight pipe is simulated with the Hydraulic Resistive Tube block. The loss due to curvature is simulated with the Local Resistance block, and the pressure loss coefficient is determined in accordance with the Crane Co. recommendations (see [1] on page 1-888, p. A-29). The flow regime is checked in the underlying Local Resistance block by comparing the Reynolds number to the specified critical Reynolds number value.

The pressure loss due to curvature for turbulent flow regime is determined according to the following formula:

$$p = K\frac{\rho}{2A^2}q\,|\,q\,|$$

where

q	Flow rate

p	Pressure loss
K	Pressure loss coefficient
A	Bend cross-sectional area
ρ	Fluid density

For laminar flow regime, the formula for pressure loss computation is modified, as described in the reference documentation for the Local Resistance block.

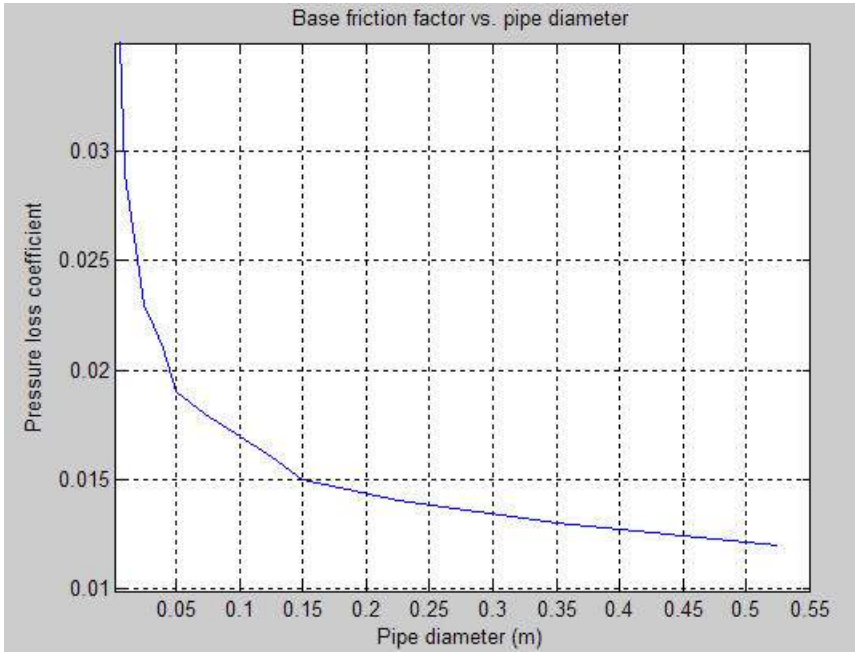
The pressure loss coefficient is determined according to recommendation provided in [1] on page 1-888:

$$K = K_d \cdot K_r \cdot K_\alpha$$

where

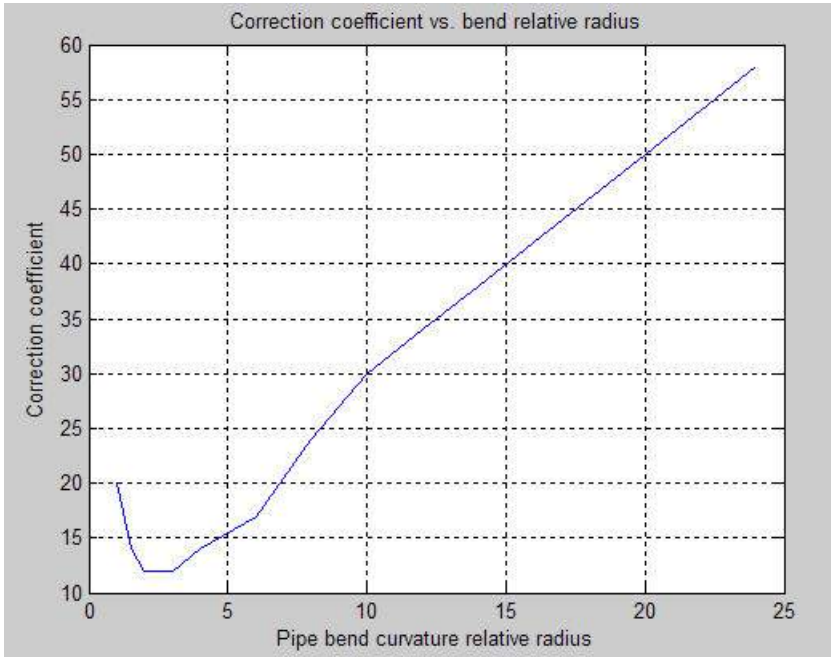
K_d	Base friction factor coefficient
K_r	Correction coefficient accounting for the bend curvature
K_α	Correction coefficient accounting for the bend angle

The base friction factor coefficient is determined according to the following table.



Note Friction factors for pipes with diameters greater than 525 mm are determined by extrapolation.

The correction coefficient accounting for the bend curvature is determined according to the next table.



The bend curvature relative radius is calculated as

```
r = bend radius / pipe diameter
```

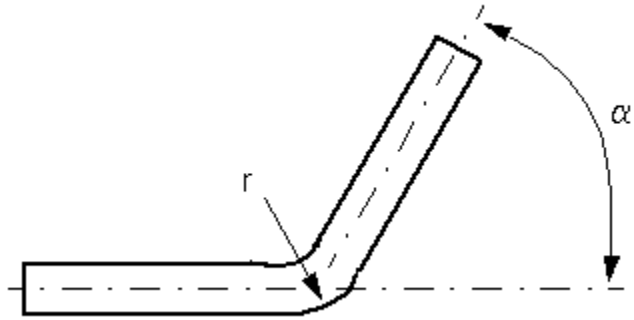
Note For pipes with the bend curvature relative radius value outside the range of $1 > r > 24$, correction coefficients are determined by extrapolation.

Correction for non-90° bends is performed with the empirical formula (see [2] on page 1-888, Fig. 4.6):

$$K_{\alpha} = \alpha(0.0142 - 3.703 \cdot 10^{-5} \alpha)$$

where

α	Bend angle in degrees ($0 \le \alpha \le 180$)



Connections A and B are conserving hydraulic ports associated with the block inlet and outlet, respectively.

The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure differential is determined as

$p = p_A - p_B$.

Warning The formulas used in the Pipe Bend block are very approximate, especially in the laminar and transient flow regions. For more accurate results, use a combination of the Local Resistance block with a table-specified *K=f(Re)* relationship and the Hydraulic Resistive Tube block.

Basic Assumptions and Limitations

- Fluid inertia and wall compliance are not taken into account.
- The bend is assumed to be made of a clean commercial steel pipe.

Parameters

Pipe diameter

The internal diameter of the pipe. The default value is `0.01` m.

Bend radius

The radius of the bend. The default value is `0.04` m.

Bend angle

The angle of the bend. The value must be in the range between 0 and 180 degrees. The default value is `90` deg.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `350`.

Fluid compressibility

Dynamic compressibility setting. Select `On` to make the fluid density dependent on pressure and temperature. Select `Off` to treat the fluid density as a constant. Dynamic compressibility impacts the transient response of the fluid at small time scales and can slow down simulation.

Initial liquid pressure (gauge)

Gauge pressure in the pipe bend at time zero. The default value is `0` Pa.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the bend inlet.

B

Hydraulic conserving port associated with the bend outlet.

References

[1] *Flow of Fluids Through Valves, Fittings, and Pipe*, Crane Valves North America, Technical Paper No. 410M

[2] George R. Keller, *Hydraulic System Analysis*, Published by the Editors of Hydraulics & Pneumatics Magazine, 1970

See Also

Elbow | Gradual Area Change | Hydraulic Resistive Tube | Local Resistance | Sudden Area Change | T-junction

Introduced in R2006b

Pipe (TL)

Closed conduit for the transport of fluid between thermal liquid components
Library: Simscape / Fluids / Thermal Liquid / Pipes & Fittings

Description

The Pipe (TL) block models the flow of a thermal liquid through a closed conduit such as a pipe. The wall of the conduit can be rigid or flexible, the latter case allowing for expansion and contraction in the radial direction, as a compliant hose or, in the life sciences, an artery might. The thermal liquid ports can be at different elevations and the vertical distance between them can be controlled (via physical signal), for example to capture the banking of an aircraft changing course.

The pressure loss across the pipe is determined as a function of friction between the fluid and the pipe and of the rise or drop in height between the ports. For enhanced precision in models with rapid flow changes (such as those associated with the water hammer effect) the block can be configured to capture the dynamic compressibility of the fluid and its inertia. Note that such effects can reduce the speed of simulation and should be used only when necessary.

The temperature change across the pipe is determined from the energy exchanges between the pipe and the remainder of the model. These exchanges include those attributed to advection and conduction of internal energy through the thermal liquid ports

(**A** and **B**) and to convection of heat through the thermal port (**H**). The calculation captures also the differences in elevation and static pressure established during simulation between the thermal liquid ports.

If the thermal liquid is treated as compressible, the pipe can be discretized into equal segments, each containing a portion of the total fluid volume. Internal fluid volumes such as these serve a special purpose in the thermal liquid domain: they provide the computational nodes at which to compute the domain and component variables during simulation. The greater the number of pipe segments, the finer the discretization, and the more accurate the simulation results (albeit at reduced simulation speed).

Block Variants

To capture the rise in port elevation, the block provides two variants. The default option treats this quantity as a constant (specified via the **Elevation gain from port A to port B** block parameter). The alternative variant treats it as a variable (controlled by physical signal port **El**). To change block variants, right-click the block and, from its context-sensitive menu, select **Simscape** + **Block Choices**. Click the desired variant: `Constant elevation` or `Variable elevation`.

Parameterizations

To model the friction losses using the data best suited for a particular application, the block provides an array of friction parameterizations. Some are based on analytical expressions requiring only a small number of empirical constants; the Haaland correlation is one such expression. Others are based on tabulated data relating various quantities of interest—the Darcy friction factor against the Reynolds number, for example, or the nominal pressure drop to the nominal mass flow rate.

Analytical and tabulated parameterizations are provided also for the heat transfer between the thermal liquid and the pipe wall. Analytical parameterizations include those based on the empirical correlations of Gnielinski and Dittus-Boelter. Tabulated parameterizations include those based on data relating the Colburn factor to the Reynolds number, the Nusselt number to the Reynolds and Prandtl numbers, or the nominal temperature differential to the nominal mass flow rate.

Discretizing the Pipe

If the pipe is segmented so that it contains more than one fluid volume, then the total mass, momentum, and energy accumulation within its span are determined as sums over

the volumes that the pipe contains. Segmented pipes are treated as assemblies of smaller pipes, each pipe associated with a separate instance of this block (each block configured to provide one fluid volume). The calculations described for this block apply to a pipe with a single fluid volume.

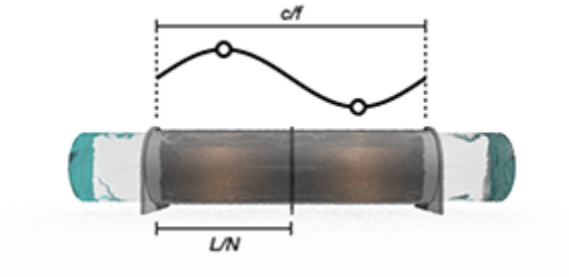
The appropriate number of pipe segments to use in a model depends partly on the time scales over which temperature and pressure disturbances tend to propagate through the pipe. Pressure waves travel the fastest (they do so at the speed of sound in the fluid) and are often the rate-limiting factor to consider. In accordance with the Nyquist sampling theorem, in order to capture an elementary sinusoidal disturbance, at least two computational nodes—and therefore pipe segments—must be available for sampling within one wavelength:

$$\frac{c}{f} = 2\frac{L}{N},$$

where c is the speed of sound, f is the frequency of the disturbance (in Hertz), L is the total length of the pipe, and N is the number of pipe segments. The left-hand side represents the wavelength of the pressure disturbance and the right-hand side the length of a pipe segment—each providing one computational node to the pipe. To capture those pressure disturbances with frequencies up to a maximum f_{Max}, the number of segments in the pipe must therefore be at least:

$$N = 2L\frac{f}{c},$$

Use this expression as a loose guideline in setting the discretization of the pipe. Other modeling constraints may factor into the decision of how many pipe segments to use and even of how to model them. More pipe segments may be required, for example, to properly define a thermal boundary condition along the length of the pipe; the pipe segments are in such a case more adequately modeled explicitly, using a separate Pipe (TL) block for each (and employing its thermal port to set the thermal boundary condition).



Use Simscape data logging to access the thermal liquid properties and states at the various nodes corresponding to the pipe segments.

Mass Balance

The thermal liquid flow enters and exits the pipe via thermal liquid ports **A** and **B**. In the default case of a rigid pipe, the volume of fluid contained between these ports is fixed. If the thermal liquid is treated as incompressible, its density (at a given operating condition) is fixed also and its mass within the pipe cannot vary with time. The mass balance between the ports is, in this simple case:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ denotes a mass flow rate into the pipe and the subscript denotes the port at which its value is defined. If the pipe is given a radially compliant wall—that is, if the **Pipe wall specification** block parameter is changed to `Flexible`—the thermal liquid mass contained within its bounds is free to vary, in a measure always directly proportional to the volume of the pipe:

$$\dot{m}_A + \dot{m}_B = \rho_I \dot{V},$$

where ρ is the thermal density inside the pipe volume (subscript `I`), denoted V. If in addition the thermal liquid is made compressible—if the **Fluid dynamic compressibility** block parameter is changed to `On`—its mass within the pipe must change with pressure and temperature also. This dependence is captured by the bulk modulus and thermal expansion coefficient of the thermal liquid:

$$\dot{m}_\mathrm{A} + \dot{m}_\mathrm{B} = \rho_\mathrm{I}\dot{V} + \rho_\mathrm{I}V\left(\frac{\dot{p}_\mathrm{I}}{\beta_\mathrm{I}} - \alpha_\mathrm{I}\dot{T}_\mathrm{I}\right),$$

where p and T denote the pressure and temperature within the pipe volume and β and α denote the bulk modulus and thermal expansion coefficient (as provided to the thermal liquid network by the Thermal Liquid Settings (TL) or Thermal Liquid Properties (TL) block).

Momentum Balance

The thermal liquid flow is subjected to different forces as it traverses the pipe. These arise due to static pressure at the ports, viscous friction along the pipe wall, and gravity on the volume of the pipe. The inertial force on the fluid is by default ignored, a suitable approximation at the large time scales over which changes to flow typically occur. The pipe is then treated as a quasi-steady component and its momentum balance—expressed as a pressure difference between its ports—becomes:

$$p_\mathrm{A} - p_\mathrm{B} = p_\mathrm{F,A} - p_\mathrm{F,B} + \rho_\mathrm{I}\, g\Delta z,$$

where p is the pressure at a port and p_F the pressure loss due to friction in half of the pipe volume; g is the gravitational acceleration and Δz the rise in elevation from port **A** to port **B**. The subscripts denote a port (**A** or **B**) or the computational node corresponding to the internal fluid volume (**I**). Pressure is assumed in this simple case to vary linearly between the ports. Its value at the internal node—that used in the table lookup of ρ_I—is for this reason defined as the arithmetic mean of the pressures at the ports:

$$p_\mathrm{I} = \frac{p_\mathrm{A} + p_\mathrm{B}}{2}.$$

If the flow is treated as compressible—if the **Fluid dynamic compressibility** block parameter is changed to `On`—the pressure in the pipe can vary nonlinearly between the ports. Its value is no longer a simple arithmetic mean and it must be obtained explicitly by another means. To carry out this calculation, the momentum balance is split over two control volumes, one each for half of the pipe volume. Between port **A** and the internal node:

$$p_\mathrm{A} - p_\mathrm{I} = p_\mathrm{F,A} + \rho_\mathrm{I}\, g\frac{\Delta z}{2}.$$

Between port **B** and the internal node:

$$p_B - p_I = p_{F,B} - \rho_I g \frac{\Delta z}{2}.$$

If, in addition, the inertia of the fluid is factored into the calculations—that is, if the **Fluid inertia** block parameter is set to `On`—then changes to flow momentum are no longer assumed to be instantaneous. The transient phase between old and new steady states becomes gradual, with a short but nonzero time scale that depends in part on the system modeled. The momentum balance becomes, in the control volume adjacent to port **A**:

$$p_A - p_I = p_{F,A} + \rho_I g \frac{\Delta z}{2} + \frac{\dot{m}_A}{S}\frac{L}{2},$$

where L is the length of the pipe and S is the cross-sectional area of the flow through the same. Reversing the sign of the elevation term gives for the control volume adjacent to port **B**:

$$p_B - p_I = p_{F,B} - \rho_I g \frac{\Delta z}{2} + \frac{\dot{m}_B}{S}\frac{L}{2},$$

Frictional Pressure Loss at the Pipe Wall

The calculation of the major pressure loss (due to friction in the pipe) varies with the viscous friction parameterization. For all parameterizations but `Nominal pressure drop vs. nominal mass flow rate`, the calculation is based on the Darcy-Weisbach equation:

$$p_{F,j} = f_{D,j}\frac{1}{4}\frac{L_E \dot{m}_j \left|\dot{m}_j\right|}{\rho_I D S^2},$$

where f_D is the Darcy friction factor and the subscript j denotes the pipe half—that adjacent to port **A** or to port **B**. L_E is the effective length of the pipe and D is the hydraulic diameter of the same. The effective pipe length is as the sum of the true pipe length and the aggregate equivalent length of all local resistances (those due to elbows, unions, fittings, and other local sources of friction).

When the flow is laminar, the friction factor (for a given pipe geometry) is a function of the Reynolds number alone:

$$f_{\mathrm{D},j} = \frac{\lambda}{\mathrm{Re}_j},$$

where λ is the shape factor of the pipe, an empirical constant used to encode the effect of pipe geometry on the major pressure loss; its value is `64` in circular pipes and `48–96` in noncircular ones. The Reynolds number at port k is defined as:

$$\mathrm{Re}_j = \frac{\dot{m}_j D}{\mu_{\mathrm{I}} S},$$

where μ is the dynamic viscosity obtained from the Thermal Liquid Settings (TL) or Thermal Liquid Properties (TL) block. The actual pressure loss calculation in the laminar flow regime is carried out as:

$$p_{\mathrm{F},j} = \frac{1}{4}\frac{\lambda \mu_{\mathrm{I}} L_{\mathrm{E}} \dot{m}_j}{\rho_{\mathrm{I}} D^2 S},$$

When the flow is turbulent, the friction factor is a function also of the pipe diameter and surface roughness. If the viscous friction parameterization is set to `Haaland correlation`, the friction factor is calculated from the empirical expression:

$$\frac{1}{\sqrt{f_{\mathrm{D},j}}} = -1.8\log\left[\left(\frac{\varepsilon / D}{3.7}\right)^{1.11} + \frac{6.9}{\mathrm{Re}_j}\right],$$

where ε is the absolute roughness of the pipe, a measure of the height of the bumps at the pipe-fluid interface; typical roughness values range from 0.0015 mm for certain plastic and glass tubes to 3 mm for larger concrete pipes. If the viscous friction parameterization is set to `Tabulated data – Darcy friction factor vs. Reynolds number`, the friction factor is obtained from the tabulated data as a function of the Reynolds number:

$$f_{\mathrm{D}} = f(\mathrm{Re}).$$

If the viscous friction parameterization is set to `Nominal pressure drop vs. nominal mass flow rate`, the major pressure loss is calculated for each pipe half from the expression:

$$p_{\mathrm{F},j} = \frac{1}{2} K_{\mathrm{p}} \dot{m}_j \sqrt{\dot{m}_j^2 + \dot{m}_{\mathrm{Th}}^2}$$

where $\dot{m}_{\mathrm{Th}}$ is a threshold mass flow rate, a small value specified in the block dialog box that is used for numerical smoothing purposes; K_{p} is a pressure loss coefficient, computed for rigid pipes as:

$$K_{\mathrm{p}} = \frac{p_{\mathrm{F,N}}}{\dot{m}_{\mathrm{N}}^2},$$

where the subscript N denotes a value specified at some nominal operating conditions. The nominal pressure and mass flow rate can be specified as scalars (in which case the pressure loss coefficient is fixed throughout simulation) or as a vector (in which case the pressure loss coefficient is determined as a variable, by interpolation or extrapolation of the tabulated data). The pressure loss coefficient is redefined for flexible pipes (with a slight change in physical units) as:

$$K_{\mathrm{p}} = \frac{p_{\mathrm{F,N}}}{\dot{m}_{\mathrm{N}}^2} D_{\mathrm{N}}.$$

The hydraulic diameter is a measure of width for pipes whose cross sections may not be circular. Note that only rigid pipes are allowed to be noncircular in cross section. The diameter of a flexible pipe can vary during simulation as a function of pressure, with such changes assumed to be uniform throughout the length of the pipe. The deformation of the pipe proceeds at a rate set in part by a viscoelastic time constant:

$$\dot{D} = \frac{D_{\mathrm{S}} - D_{\mathrm{N}}}{\tau},$$

where τ is the time constant and the subscripts S and N denote the values at steady-state and nominal conditions, respectively. The nominal value gives the diameter at zero gauge pressure (when the pressure in the component is equal to atmospheric pressure). The steady-state value gives the diameter at the actual gauge pressure after the transient response has ceased:

$$D_{\mathrm{S}} = K_c \frac{p - p_0}{\tau},$$

where K_c is the elastic compliance of the pipe wall, a number indicating the extent to which a change in pressure affects the diameter of the pipe. This parameter can be calculated if necessary from other elastic properties of the wall:

$$K_c = \frac{D_{0,\mathrm{Int}}}{E}\left(\frac{D_{0,\mathrm{Ext}}^2 + D_{0,\mathrm{Int}}^2}{D_{0,\mathrm{Ext}}^2 - D_{0,\mathrm{Int}}^2} + \nu\right),$$

where E and ν are the modulus of elasticity and Poisson ratio of the pipe wall material. The subscript `0` denotes an initial value, corresponding to the conditions in the model at the start of simulation (not to be confused with nominal conditions at which D_N is specified). The subscripts `Int` and `Ext` refer to the internal and external circumferences of the pipe wall.

The nominal pipe diameter (used in the calculation of the pipe deformation rate) is computed as:

$$D_N = \sqrt{\frac{4S}{\pi}},$$

where S is the specified cross-sectional area of the pipe (a nominal value in pipes that are treated as flexible).

Energy Balance

The energy of the fluid in the pipe can change by a variety of processes. These include advection and conduction through the ends of the pipe (thermal liquid ports **A** and **B**), convection at the pipe-fluid interface (thermal port **H**), and, in pipes that are set at an angle, longitudinal changes in elevation. Expressing the energy balance in terms of the energy accumulation rate in the pipe gives:

$$\dot{E}_I = \phi_A + \phi_B + \phi_H - \dot{m}_I\, g\Delta z,$$

where $\dot{E}$ is the energy accumulation rate and ϕ is the energy flow rate through a port—smoothed and upwinded in the thermal liquid ports, as described in “Energy Flows in Thermal Liquid Networks”. As in the mass and momentum calculations, the subscript I denotes a value defined at the internal computational node. The mass flow rate in the potential energy term is the average of those established at the thermal liquid ports:

$$\dot{m}_{\mathrm{I}} = \frac{\dot{m}_{\mathrm{A}} + \dot{m}_{\mathrm{B}}}{2}.$$

The total energy of the internal fluid volume is defined in terms of the specific internal energy as:

$$E_{\mathrm{I}} = \rho_{\mathrm{I}} u_{\mathrm{I}} V,$$

where u is the specific internal energy of the fluid, obtained as a function of temperature and pressure from the Thermal Liquid Settings (TL) or Thermal Liquid Properties (TL) block, and V is the internal volume of the pipe. If the flow is treated as compressible—if the **Flow dynamic compressibility** block parameter is set to `On`—then the energy accumulation rate in the pipe is computed as:

$$\dot{E}_{\mathrm{I}} = \rho_{\mathrm{I}} V \left(\frac{du}{dp}\frac{dp}{dt} + \frac{du}{dT}\frac{dT}{dt} \right)_{\mathrm{I}},$$

If in addition the pipe is given a compliant wall—if the **Pipe wall specification** block parameter is set to `Flexible`—then the volume of thermal liquid within its bounds is free to vary. The energy accumulation rate becomes:

$$\dot{E}_{\mathrm{I}} = \rho_{\mathrm{I}} V \left(\frac{du}{dp}\frac{dp}{dt} + \frac{du}{dT}\frac{dT}{dt} \right)_{\mathrm{I}} + \left(\rho_{\mathrm{I}} u_{\mathrm{I}} + p_{\mathrm{I}} \right) \left(\frac{dV}{dt} \right)_{\mathrm{I}},$$

The heat flow rate between the thermal liquid and the pipe wall is assumed to result from a convective exchange and a purely conductive exchange:

$$\phi_{\mathrm{H}} = Q_{\mathrm{Conv}} + Q_{\mathrm{Cond}}.$$

The heat flow rate due to conduction is computed as:

$$Q_{\mathrm{Cond}} = \frac{k_{\mathrm{I}} S_{\mathrm{H}}}{D} \left(T_{\mathrm{H}} - T_{\mathrm{I}} \right),$$

where k is the thermal conductivity of the thermal liquid and S_{H} is the surface area of the pipe wall (the product of the perimeter and length of the pipe, not to be confused with the cross-sectional area of the same). The subscripts `H` and `I` denote the pipe wall and the internal fluid volume, respectively.

The heat flow rate due to convection is computed as:

$$Q_{\mathrm{Conv}} = c_{\mathrm{p,Avg}} \left|\dot{m}_{\mathrm{Avg}}\right| \left(T_{\mathrm{H}} - T_{\mathrm{In}}\right) \left[1 - \exp\left(\frac{hS_{\mathrm{H}}}{c_{\mathrm{p,Avg}} \left|\dot{m}_{\mathrm{Avg}}\right|}\right)\right],$$

where c_{p} is the specific heat of the thermal liquid, $_{\mathrm{h}}$ is the heat transfer coefficient of the pipe. The subscript In denotes the pipe inlet (port **A** or **B** depending on flow direction). Parameters with the subscript Avg are evaluated at the average temperature of the pipe. This expression is based on the assumption that temperature varies exponentially between the ends of the pipe.

For all heat transfer parameterizations but `Nominal temperature differential vs. nominal mass flow rate`, the heat transfer coefficient is computed from the expression:

$$h = \frac{\mathrm{Nu}_{\mathrm{Avg}} k_{\mathrm{Avg}}}{D},$$

where *Nu* is the Nusselt number and *k* the thermal conductivity in the pipe, both obtained at the average temperature inside it. The Nusselt number calculation varies with the parameterization selected:

- `Gnielinski correlation`:

  In the turbulent regime:

  $$\mathrm{Nu}_{\mathrm{Avg}} = \frac{f_{\mathrm{Avg}}/8 \left(\mathrm{Re}_{\mathrm{Avg}} - 1000\right)\mathrm{Pr}_{\mathrm{Avg}}}{1 + 12.7\left(f_{\mathrm{Avg}}/8\right)^{1/2}\left(\mathrm{Pr}_{\mathrm{Avg}}^{2/3} - 1\right)},$$

  where *Pr* is the Prandtl number. In the laminar flow regime, in which the correlation does not apply, the Nusselt number is obtained as a constant (denoted Nu_{L}) from the **Nusselt number for laminar flow heat transfer** block parameter:

  $$\mathrm{Nu}_{\mathrm{Avg}} = \mathrm{Nu}_{\mathrm{L}},$$

- `Dittus-Boelter correlation`:

In the turbulent regime:

$$\mathrm{Nu}_{\mathrm{Avg}} = a\mathrm{Re}_{\mathrm{Avg}}^{b}\mathrm{Pr}_{\mathrm{Avg}}^{c},$$

where a, b, and c are empirical constants specific to the system considered. The default values specified in the block are those used in the exact form of the Dittus-Boelter expression for a fluid being warmed by the pipe wall:

$$\mathrm{Nu}_{\mathrm{Avg}} = 0.023\mathrm{Re}_{\mathrm{Avg}}^{0.8}\mathrm{Pr}_{\mathrm{Avg}}^{0.4}.$$

As with the Gnielinski correlation, in the laminar flow regime, in which the correlation does not apply, the Nusselt number is obtained as a constant (denoted Nu_{L}) from the **Nusselt number for laminar flow heat transfer** block parameter.

- `Tabulated data - Colburn factor vs. Reynolds number`:

  In all flow regimes:

  $$\mathrm{Nu}_{\mathrm{Avg}} = \mathrm{J}_{\mathrm{M,Avg}}(\mathrm{Re}_{\mathrm{Avg}})\mathrm{Re}_{\mathrm{Avg}}\mathrm{Pr}_{\mathrm{Avg}}^{1/3}.$$

  where J_{M} is the Colburn-Chilton factor.

- `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`:

  In all flow regimes:

  $$\mathrm{Nu}_{\mathrm{Avg}} = \mathrm{Nu}(\mathrm{Re}_{\mathrm{Avg}}, \mathrm{Pr}_{\mathrm{Avg}}).$$

The calculations differ slightly in the case of the `Nominal temperature difference vs. nominal mass flow rate` parameterization. In the laminar flow regime, the heat transfer coefficient is the same constant specified in other parameterizations (**Nusselt number for laminar flow heat transfer** block parameter). In the turbulent flow regime, it is calculated as a function of mass flow rate, with the proportionality between the two fixed by a form of the Dittus-Boelter correlation:

$$\mathrm{Nu} \propto \mathrm{Re}^{0.8},$$

or:

$$\frac{hD}{k} \propto \left(\frac{\dot{m}D}{S\mu} \right)^{0.8}$$

Rearranging terms:

$$h = K_{\mathrm{H,Avg}} \frac{\dot{m}_{\mathrm{Avg}}^{0.8}}{D^{1.8}},$$

where K_{H} is a proportionality constant created by lumping all parameters but those retained in the final expression (with the fluid properties defined at the average temperature in the pipe). The constant is computed from nominal values obtained for h, D, and $\dot{m}$ as:

$$K_{\mathrm{H,Avg}} = \frac{h_{\mathrm{N}} D_{\mathrm{N}}^{1.8}}{\dot{m}_{\mathrm{N}}^{0.8}},$$

The heat transfer coefficient for the `Nominal temperature difference vs. nominal mass flow rate` parameterization is therefore:

$$h = \frac{h_{\mathrm{N}} D_{\mathrm{N}}^{1.8}}{\dot{m}_{\mathrm{N}}^{0.8}} \frac{\dot{m}_{\mathrm{I}}^{0.8}}{D^{1.8}},$$

or, in the simpler case of pipe treated as rigid (and therefore assumed to be constant in diameter):

$$h_{\mathrm{I}} = \frac{h_{\mathrm{N}}}{\dot{m}_{\mathrm{N}}^{0.8}} \dot{m}_{\mathrm{I}}^{0.8}.$$

The nominal mass flow rate is obtained from the tabulated data specified via the **Nominal mass flow rate** block parameter. The nominal heat transfer coefficient is calculated from various nominal parameters as:

$$h_{\mathrm{N}} = \frac{\dot{m}_{\mathrm{N}} c_{\mathrm{p,N}}}{S_{\mathrm{H,N}}} \ln\left(\frac{T_{\mathrm{H,N}} - T_{\mathrm{In,\,N}}}{T_{\mathrm{H,N}} - T_{\mathrm{Out,N}}} \right),$$

where c_p is the specific heat at constant pressure and the subscripts `H`, `In`, and `Out` denote the wall, the inlet (whichever of the thermal liquid ports happens to be it at a given moment), and the outlet. The nominal surface area of the pipe wall ($S_{\{H,N\}}$) is computed as the product of the pipe circumference and the pipe length:

$$S_{H,N} = \sqrt{\frac{4S}{D}}L,$$

The hydraulic diameter (D) is a constant if the pipe is rigid but a function of pressure if the pipe is flexible. Its value is obtained from the **Hydraulic diameter** block parameter if the **Pipe wall specification** parameter is set to `Rigid` and computed from the **Nominal cross-sectional area** parameter otherwise, giving, for the flexible pipe:

Ports

Input

`El` — Port elevation difference

physical signal

Control signal with which to set the instantaneous elevation difference between the thermal liquid ports. Depending on the blocks used to generate the signal, the elevation difference can be constant or it can vary with time. If the port is left unconnected, the elevation difference is fixed at zero during simulation.

Conserving

`A` — Pipe opening

thermal liquid

Opening through which the thermal liquid flows into or out of the pipe. Ports **A** and **B** can each function as either inlet or outlet. Thermal conduction is allowed between the thermal liquid ports and the fluid internal to the pipe (though its impact is typically relevant only at near zero flow rates).

`B` — Pipe opening

thermal liquid

Opening through which the thermal liquid flows into or out of the pipe. Ports **A** and **B** can each function as either inlet or outlet. Thermal conduction is allowed between the thermal liquid ports and the fluid internal to the pipe (though its impact is typically relevant only at near zero flow rates).

`H` — Pipe wall
thermal

Thermal boundary between the fluid volume and the pipe wall. Use this port to capture heat exchanges of various kinds—for example, conductive, convective, or radiative—between the fluid and the environment external to the pipe (taking into account the thermal resistance of the wall when it is significant).

Parameters

Configuration

`Fluid dynamic compressibility` — Option to capture variations in density due to pressure
`On` (default) | `Off`

Option to capture variations in density due to those in pressure. Dynamic compressibility lends the block to more realistic, though also more computationally demanding, simulations. Select `Off` if simulation speed is of concern, as it generally is in real-time simulation, and if dynamic compressibility is of little consequence in the results of the model.

`Fluid inertia` — Option to capture the resistance of the fluid to changes in its motion
`On` (default) | `Off`

Option to capture the resistance of the fluid to changes in its state of motion, for example in systems prone to water hammer. Fluid inertia lends the block to more realistic, though also more computationally demanding, simulations. Select `Off` if simulation speed is of concern, as it generally is in real-time simulation, and if fluid inertia is of little consequence in the results of the model.

Dependencies

This parameter is active when the **Fluid dynamic compressibility** block parameter is set to `On`.

`Number of segments` — Number of lengths into which to discretize the pipe
`1` (default) | positive unitless scalar

Number of lengths into which to discretize the pipe. Each length corresponds to a fluid volume with a computational node—the point in a component at which pressure and temperature are evaluated during simulation. Increase the number of pipe segments to more accurately capture the lengthwise distributions of pressure and temperature, for example in the simulation of water hammer, in which such distributions matter.

Dependencies

This parameter is active when the **Fluid dynamic compressibility** block parameter is set to `On`.

`Pipe total length` — Distance between the ports of the pipe
`5 m` (default) | positive scalar in units of length

Sum of the lengths of the segments comprising the pipe.

`Nominal cross-sectional area` — Area of the inner circumference of the cross section of the pipe
`0.01` m^2 (default) | positive scalar in units of area

Area of the inner circumference of the cross section of the pipe (in the undeformed state if modeled with a compliant wall).

`Pipe wall specification` — Option to capture the radial compliance of the pipe wall
`Rigid` (default) | `Flexible`

Option to capture the radial compliance of the pipe wall. The default setting of `On` corresponds to a flexible tube whose wall expands and contracts as a function of pressure. Select `Off` if simulation speed is of concern, as it generally is in real-time simulation, and if wall compliance is of little consequence in (or uncharacteristic of) the model.

Dependencies

This parameter is active when the **Fluid dynamic compressibility** block parameter is set to `On`.

`Hydraulic diameter` — Ratio of the pipe opening area to the perimeter of the same

`0.1128 m` (default) | positive scalar in units of length

Ratio of the opening area of the pipe to the inner perimeter of (the cross-section of) the same. This parameter gives a general measure of width for pipes with noncircular cross sections.

Dependencies

This parameter is active when the **Fluid dynamic compressibility** block parameter is set to `Off` or when it is set to `On` but the **Pipe wall specification** block parameter is set to `Rigid`.

`Elevation gain from port A to port B` — Change in pipe elevation in the direction of port A to port B

`0 m` (default) | positive scalar in units of length

Change in the elevation of the pipe in the direction of port **A** to port **B**. This parameter allows for the calculation of the pressure change due to elevation in the pipe. The default value of `0` corresponds to a pipe laid flat.

Dependencies

This parameter is exposed in the block dialog box when the block variant is set to `Constant elevation`. Change the block variant to `Variable elevation` if necessary, for example to capture the tilting of the pipe in during simulation.

`Gravitational acceleration` — Acceleration due to gravity at the mean elevation of the pipe

`9.81 m/s^2` (default) | positive scalar in units of length/time^2

Value of the gravitational acceleration (g) at the mean elevation of the pipe. Any changes in elevation are assumed to be sufficiently small that the pull of gravity is approximately constant.

`Static pressure-diameter compliance` — Measure of the deformation induced in the pipe by a unit change in pressure

`1.2e-6 m/MPa` (default) | positive scalar in units of length/pressure

Measure of the radial deformation induced in a pipe by a unit change in the pressure within it relative to its surroundings. This parameter is a property of the material of which the pipe wall is made.

Dependencies

This parameter is active when the **Pipe wall specification** block parameter is set to `Flexible`.

`Viscoelastic process time constant` — Measure of the time over which the pipe is expected to deform

`0.01 s` (default) | positive scalar in units of time

Characteristic time scale of the elastic deformations produced on the pipe wall. This parameter gives a rough measure of the time needed for a pipe disturbed by a pressure change to reach a new steady-state diameter.

Dependencies

This parameter is active when the **Pipe wall specification** block parameter is set to `Flexible`.

Viscous Friction

`Viscous friction parameterization` — Method by which to capture the pressure loss due to friction against the pipe wall

`Haaland correlation` (default) | `Nominal pressure drop vs. nominal mass flow rate` | `Tabulated data - Darcy friction factor vs. Reynolds number`

Method by which to capture the pressure loss in the pipe due to friction against the wall. The calculation can be based on an empirical correlation (that of Haaland) or on a tabulated function (providing either the pressure drop or the Darcy friction factor).

`Aggregate equivalent length of local resistances` — Minor pressure loss in the pipe expressed as a length

`1 m` (default) | positive scalar in units of length

Minor pressure loss in the pipe expressed as a length. This parameter serves to adjust the effective length of the pipe and from it to calculate the total pressure loss between the ports.

Dependencies

This parameter is active when the **Viscous friction parameterization** block parameter is set to `Haaland correlation`.

`Internal surface absolute roughness` — Characteristic height of the microscopic protrusions on the inner surface of the pipe
`1.5e-5 m` (default) | positive scalar in units of length

Characteristic height of the microscopic protrusions on the inner surface of the pipe. This parameter serves to calculate the pressure loss due to friction against the wall of the pipe.

Dependencies

This parameter is active when the **Viscous friction parameterization** block parameter is set to `Haaland correlation`.

`Shape factor for laminar flow viscous friction` — Empirical measure of the effects of geometry on the pressure losses due to friction
64 (default) | positive unitless scalar

Empirical measure of the effects of geometry on the pressure losses due to friction. Typical values range from 48 to 96. The default value, 64, corresponds to a pipe of circular cross section.

`Laminar flow upper Reynolds number limit` — Reynolds number below which the flow is laminar
`2e+3` (default) | positive unitless scalar

Reynolds number below which the flow is laminar. Above this threshold, the flow transitions to turbulent, reaching the true turbulent regime at the **Turbulent flow lower Reynolds number limit** setting.

`Turbulent flow lower Reynolds number limit` — Reynolds number above which the flow is turbulent
`4e+3` (default) | positive unitless scalar

Reynolds number above which the flow is turbulent. Below this threshold, the flow gradually transitions to laminar, reaching the true laminar regime at the **Laminar flow upper Reynolds number limit** setting.

`Nominal mass flow rate` — Mass flow rate(s) from which to calculate the pressure loss coefficient of the pipe
`[0.1 1] kg/s` (default) | scalar or vector of numbers in units of mass/time

Mass flow rate, at some chosen operating points, from which to calculate the pressure loss coefficient of the pipe. This parameter can be scalar or vector, with a scalar

corresponding to a single operating point and a vector to a multitude of operating points. The MATLAB `mldivide` function is used to solve for the pressure loss coefficient if a vector is specified. All nominal parameters in the **Viscous Friction** tab must have the same size.

Dependencies

This parameter is active when the **Viscous friction parameterization** block parameter is set to `Nominal pressure drop vs. nominal mass flow rate`.

`Nominal pressure drop` — Pressure drop(s) from which to calculate the pressure loss coefficient of the pipe

`[0.001 0.01] MPa` (default) | scalar or vector of numbers in units of pressure

Pressure drop, at some chosen operating points, from which to calculate the pressure loss coefficient of the pipe. This parameter can be scalar or vector, with a scalar corresponding to a single operating point and a vector to a multitude of operating points. The MATLAB `mldivide` function is used to solve for the pressure loss coefficient if a vector is specified. All nominal parameters in the **Viscous Friction** tab must have the same size.

Dependencies

This parameter is active when the **Viscous friction parameterization** block parameter is set to `Nominal pressure drop vs. nominal mass flow rate`.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to apply numerical smoothing to the block calculations

`1e-6 kg/s` (default) | positive scalar in units of mass/time

Mass flow rate below which to apply numerical smoothing to the block calculations, a measure taken to prevent simulation errors due to discontinuities at zero flow.

Dependencies

This parameter is active when the **Viscous friction parameterization** block parameter is set to `Nominal pressure drop vs. nominal mass flow rate`.

`Reynolds number vector for turbulent Darcy friction factor` — Reynolds numbers at which to tabulate the Darcy friction factor

`[ 400 1000 1.5e+3 3e+3 4e+3 6e+3 1e+4 2e+4 4e+4 6e+4 1e+5 1e+8 ]` (default) | vector of positive unitless numbers

Reynolds number at which to tabulate the Darcy friction factor. This data serves to construct a one-way lookup table from which to calculate the Darcy friction factor and ultimately the pressure loss across the pipe. The vector must increase monotonically from left to right. This and the Darcy friction factor vector must have the same size.

Dependencies

This parameter is active when the **Viscous friction parameterization** block parameter is set to `Tabulated data - Darcy friction factor vs. Reynolds number`.

`Darcy friction factor vector` — Darcy friction factors at the tabulated Reynolds numbers

`[ 0.264 0.112 0.07099999999999999 0.0417 0.0387 0.0268 0.025 0.0232 0.0226 0.022 0.0214 0.0214 ]` (default) | vector of positive unitless numbers

Darcy friction factors at the tabulated values of the Reynolds number. This data serves to construct a one-way lookup table from which to calculate the Darcy friction factor and ultimately the pressure loss across the pipe. This and the Reynolds number must have the same size.

Dependencies

This parameter is active when the **Viscous friction parameterization** block parameter is set to `Tabulated data - Darcy friction factor vs. Reynolds number`.

Heat Transfer

`Heat transfer parameterization` — Method by which to capture the convective heat transfer with the pipe wall

`Gnielinski correlation` `Nominal temperature differential vs. nominal mass flow rate` (default) | `Dittus-Boelter correlation` | `Tabulated data - Colburn factor vs. Reynolds number` | `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`

Method by which to calculate the convective heat transfer coefficient of the pipe. The calculation can be based on an empirical correlation (of which those of Gnielinski and Dittus-Boelter are options) or on a tabulated function (providing the temperature differential, the Colburn factor, or the Nusselt number).

`Nusselt number for laminar flow heat transfer` — Nusselt number to use in the heat transfer calculations for laminar flows

`3.66` (default)

Nusselt number for laminar pipe flows. This number serves to calculate the heat transfer coefficient between the pipe wall and the fluid within it. The default value of `3.66` corresponds to flow through a pipe with a circular cross section.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Gnielinski correlation`, `Nominal temperature differential vs. nominal mass flow rate`, or `Dittus-Boelter correlation`.

`Nominal mass flow rate` — Mass flow rate(s) from which to calculate the heat transfer coefficient

`[0.1 1] kg/s` (default) | scalar or vector of numbers in units of mass/time

Mass flow rates, at some chosen operating point, from which to calculate the heat transfer coefficient of the pipe. This parameter can be scalar or vector, with a scalar corresponding to a single operating point and a vector to a multitude of operating points. The MATLAB `mldivide` function is used to solve for the heat transfer coefficient if vector a vector is specified. All nominal parameters in the **Heat Transfer** tab must have the same size.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Nominal temperature differential vs. nominal mass flow rate`.

`Nominal inflow temperature` — Pipe entrance temperature(s) from which to calculate the heat transfer coefficient

`[293.15 293.15] K` (default) | scalar or vector of numbers in units of temperature

Pipe entrance temperatures, at some chosen operating points, from which to calculate the heat transfer coefficient of the pipe. This parameter can be scalar or vector, with a scalar corresponding to a single operating point and a vector to a multitude of operating points. The MATLAB `mldivide` function is used to solve for the heat transfer coefficient if a vector is specified. All nominal parameters in the **Heat Transfer** tab must have the same size.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Nominal temperature differential vs. nominal mass flow rate`.

`Nominal outflow temperature` — Pipe exit temperature(s) from which to calculate the heat transfer coefficient

`[300 300] K` (default) | scalar or vector of numbers in units of temperature

Pipe exit temperatures, at some chosen operating points, from which to calculate the heat transfer coefficient of the pipe. This parameter can be scalar or vector, with a scalar corresponding to a single operating point and a vector to a multitude of operating points. The MATLAB `mldivide` function is used to solve for the heat transfer coefficient if a vector is specified. All nominal parameters in the **Heat Transfer** tab must have the same size.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Nominal temperature differential vs. nominal mass flow rate`.

`Nominal inflow pressure` — Pipe entrance pressure(s) from which to calculate the heat transfer coefficient

`[0.101325 0.101325] MPa` (default) | scalar or vector of numbers in units of pressure

Pipe entrance pressures, at some chosen operating points, from which to calculate the heat transfer coefficient of the pipe. This parameter can be scalar or vector, with a scalar corresponding to a single operating point and a vector to a multitude of operating points. The MATLAB `mldivide` function is used to solve for the heat transfer coefficient if a vector is specified. All nominal parameters in the **Heat Transfer** tab must have the same size.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Nominal temperature differential vs. nominal mass flow rate`.

`Nominal wall temperature` — Pipe wall temperature(s) from which to calculate the heat transfer coefficient

`[303.15 303.15] K` (default) | scalar or vector of numbers in units of temperature

Pipe wall temperatures, at some chosen operating points, from which to calculate the heat transfer coefficient of the pipe. This parameter can be scalar or vector, with a scalar corresponding to a single operating point and a vector to a multitude of operating points. The MATLAB `mldivide` function is used to solve for the heat transfer coefficient if a vector is specified. All nominal parameters in the **Heat Transfer** tab must have the same size.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Nominal temperature differential vs. nominal mass flow rate`.

`Coefficient a` — Empirical constant *a* of the Dittus-Boelter correlation

`0.023` (default) | positive unitless scalar

Empirical constant *a* to use in the Dittus-Boelter correlation. The correlation gives the value of the Nusselt number from which to calculate the heat transfer coefficient in turbulent pipe flows. The default value is that most often associated with this correlation.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Dittus-Boelter correlation`.

`Exponent b` — Empirical constant *b* of the Dittus-Boelter correlation

`0.8` (default) | positive unitless scalar

Empirical constant *c* to use in the Dittus-Boelter correlation. The correlation gives the value of the Nusselt number from which to calculate the heat transfer coefficient in turbulent pipe flows. The default value is that most often associated with this correlation.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Dittus-Boelter correlation`.

`Exponent c` — Empirical constant *c* of the Dittus-Boelter correlation

`0.4` (default) | positive unitless scalar

Empirical constant *c* to use in the Dittus-Boelter correlation. The correlation gives the value of the Nusselt number from which to calculate the heat transfer coefficient in turbulent pipe flows. The default value is that most often associated with this correlation when the fluid is being warmed.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Dittus-Boelter correlation`.

`Reynolds number vector for Colburn factor` — Reynolds numbers at which to tabulate the Colburn factor
`[100 150 1000]` (default) | vector of positive unitless numbers

Reynolds numbers at which to tabulate the Colburn factor. This data serves to construct a one-way lookup table from which to calculate the Colburn factor and ultimately the heat transfer coefficient. The vector must increase monotonically from left to right. This and the Colburn number vector must have the same size.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Tabulated data - Colburn factor vs. Reynolds number`.

`Colburn factor vector` — Colburn factors at which the tabulated Reynolds numbers
`[0.019 0.013 0.002]` (default) | vector of positive unitless numbers

Colburn factors at the tabulated values of the Reynolds number. This data serves to construct a one-way lookup table from which to calculate the Colburn factor and ultimately the heat transfer coefficient. This and the Reynolds number vector must have the same size.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Tabulated data - Colburn factor vs. Reynolds number`.

`Reynolds number vector for Nusselt number` — Reynolds number at which to tabulate the Nusselt number
`[100 150 100]` (default) | vector of positive unitless numbers

Reynolds numbers at which to tabulate the Nusselt number. This data serves to construct a two-way lookup table from which to calculate the Nusselt number and ultimately the heat transfer coefficient. The vector must increase monotonically from left to right. This vector must be equal in size to the number of rows in the Nusselt number table.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`.

`Prandtl number vector for Nusselt number` — Prandtl numbers at which to tabulate the Nusselt number

`[1 10]` (default) | vector of positive unitless numbers

Prandtl numbers at which to tabulate the Nusselt number. This data serves to construct a two-way lookup table from which to calculate the Nusselt number and ultimately the heat transfer coefficient. The vector must increase monotonically from left to right. This vector must be equal in size to the number of columns in the Nusselt number table.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`.

`Nusselt number table` — Nusselt numbers at the tabulated Reynolds and Prandtl numbers

`[ 3.72 4.21; 3.75 4.44; 4.21 7.15 ]` (default) | matrix of positive unitless numbers

Nusselt numbers at the tabulated values of the Reynolds and Prandtl numbers. This data serves to construct a two-way lookup table from which to calculate the Nusselt number and ultimately the heat transfer coefficient.

The Reynolds number changes from row to row from top to bottom. The Prandlt number changes from column to column from left to right. The number of rows must be equal to the length of the Reynolds number vector and the number of columns to the length of the Prandtl number vector.

Dependencies

This parameter is active when the **Heat transfer parameterization** block parameter is set to `Tabulated data - Nusselt number vs. Reynolds number & Prandtl number`.

Initial Conditions

`Initial liquid temperature` — Absolute temperature in the pipe at the start of simulation

`293.15 K` (default) | positive scalar or vector in units of temperature

Absolute temperature in the pipe at the start of simulation. This parameter can be a scalar, a vector of two elements, or a vector equal in size to the number of segments in

the pipe. A scalar prescribes a constant temperature from end to end, a two-element vector a linear temperature gradient, and a vector of *N* values the individual temperatures of the various pipe segments.

`Initial liquid pressure` — Absolute pressure in the pipe at the start of simulation
`0.101325 MPa` (default) | positive scalar or vector in units of pressure

Pressure in the pipe at the start of simulation. This parameter can be a scalar, a vector of two elements, or a vector equal in size to the number of segments in the pipe. A scalar prescribes a constant pressure from end to end, a two-element vector a linear pressure gradient, and a vector of *N* values the individual pressures of the various pipe segments.

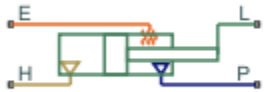
See Also

Sudden Area Change (TL)

Introduced in R2016a

Pneumo-Hydraulic Actuator

Double-acting cylinder operated by hydraulic and pneumatic power

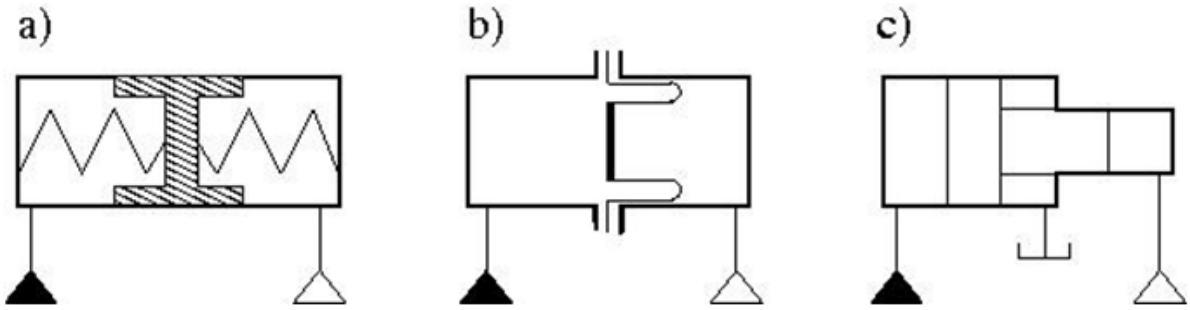


Library

Hydraulic Cylinders

Description

The Pneumo-Hydraulic Actuator block simulates a pneumo-hydraulic actuator, implemented as a double-acting cylinder with one side connected to a hydraulic power supply and another side operated by pneumatic power. Such devices are widely used as pneumo-hydraulic pumps, intensifiers, and converters of various types. The following illustration shows a few examples of the pneumo-hydraulic actuator: a) with rigid separator; b) with flexible separator; c) pneumo-hydraulic intensifier.



The block provides two main modeling variants, accessible by right-clicking the block in your block diagram and then selecting the appropriate option from the context menu, under **Simscape > Block choices**:

- **One mechanical port** — Use this variant to model just the load on the piston. In this case, the cylinder is assumed to be grounded.
- **Two mechanical ports** — Use this variant to model the load on the actuator cylinder, as well as the piston. This variant also lets you include liquid compressibility on the hydraulic side of the actuator.

Configuration with One Mechanical Port

Use this variant of the block when the load is applied only to the piston and the cylinder clamping structure is grounded. The hydraulic part of the model accounts only for fluid consumption associated with the piston velocity. The pneumatic part of the model is built with the ideal gas relationships. To simulate the limit on the piston motion, the hard stop is included in the model. The piston effective area is assumed to be constant. As a result, the model is described with the following equations:

$$q_H = A_H \cdot v$$

$$F_H = A_H \cdot p$$

$$F_P = A_P \cdot p$$

$$F_{HS} = \begin{cases} K \cdot (x - stroke) + D \cdot v & \text{for } x > stroke \\ K \cdot x + D \cdot v & \text{for } x < 0 \\ 0 & \text{for } 0 \le x \le stroke \end{cases}$$

$$F_H = F_P + F_{HS} + F_L$$

$$v = \frac{dx}{dt}$$

$$V_P = V_0 + A_P \cdot (stroke - x)$$

$$G = \frac{V_P}{RT}\left(\frac{dp}{dt} - \frac{p}{T}\frac{dT}{dt}\right) - \frac{A_P}{RT}p \cdot v$$

$$Q_P = \frac{c_v \cdot V_P}{R}\frac{dp}{dt} - \frac{c_p \cdot A_P}{R} p \cdot v + Q_{HE}$$

where

q_H	Hydraulic volumetric flow rate
p	Pressure in the actuator chambers
A_H	Hydraulic side effective area
A_P	Pneumatic side effective area
v	Piston velocity
F_H	Force developed by piston on hydraulic side
F_P	Force developed by piston on pneumatic side
F_{HS}	Hard stop force
F_L	Force developed by external load connected to port L
K	Hard stop stiffness
D	Hard stop damping
$stroke$	Piston stroke
x	Piston displacement
V_P	Chamber volume on pneumatic side
V_0	Chamber dead volume on pneumatic side
G	Gas mass flow rate
R	Gas constant
T	Gas absolute temperature
Q_P	Heat flow through the pneumatic chamber
c_v	Gas specific heat at constant volume
c_p	Gas specific heat at constant pressure
Q_{HE}	Heat flow through the thermal port E

The model is suitable for building pneumo-hydraulic or hydro-pneumatic pumps, intensifiers, and similar devices. You can simulate piston loading (such as inertia, springs, friction) by modeling the load externally and connecting it to port L. Similarly, simulate the heat exchange with the environment through the external thermal port E, which

corresponds to the gas in the chamber. Use blocks from the Simscape Foundation library, such as the Convective Heat Transfer, Conductive Heat Transfer, Thermal Mass, and so on, depending on the actual system configuration.

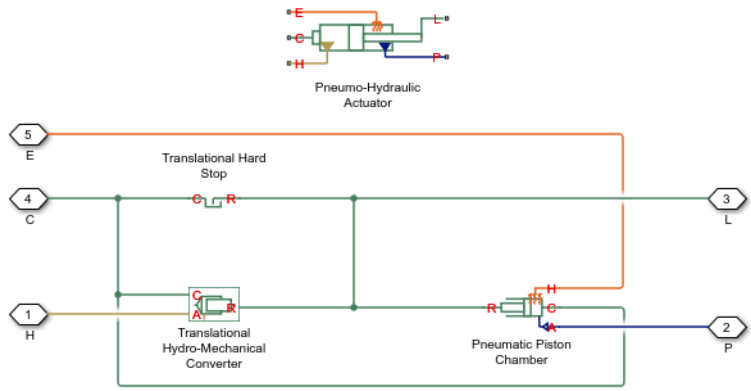
Port P is the pneumatic conserving port associated with the pneumatic side of the actuator. Port H is the hydraulic conserving port associated with the hydraulic inlet.

The block directionality assumes that pressure in the hydraulic chamber causes the piston to move in the positive direction, while pressure in the pneumatic chamber tends to move the piston in the negative direction. Flow rates are considered positive if they flow into the actuator.

Configuration with Two Mechanical Ports

This variant of the block is suitable when the forces on both piston and cylinder are considered. This is a composite component and is built of the following blocks from the Simscape Foundation library:

- Translational Hydro-Mechanical Converter
- Pneumatic Piston Chamber
- Translational Hard Stop



Connections R and C are mechanical translational ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connection H is a hydraulic conserving port that is connected to the hydraulic Translational Hydro-Mechanical Converter. Connection P is a pneumatic conserving port that is connected to the pneumatic port of the

Pneumatic Piston Chamber block. Connection E is a thermal conserving port associated with the gas in the pneumatic chamber. The fluid pressure, either on the hydraulic or pneumatic side, is transformed into mechanical energy through the converter. The piston (rod) motion is limited with the mechanical Translational Hard Stop block in such a way that the rod can travel only between cylinder caps. The fluid compressibility can also be taken into account on the hydraulic side.

You can control the block directionality using the **Converter orientation** parameter. If the **Converter orientation** is set to `Act in positive direction`, the pressure in the hydraulic chamber causes the piston to move in the positive direction, while the pressure in the pneumatic chamber tends to move the piston in the negative direction. Flow rates are considered positive if they flow into the actuator.

Basic Assumptions and Limitations

- The effective piston area in each chamber is assumed to be constant.
- The leakage flow between chambers is assumed to be negligible because pressures in the chambers are equal.
- In **One mechanical port** configuration, on the hydraulic side, fluid compressibility is not taken into account. However, in **Two mechanical ports** configuration, you have an option to include fluid compressibility by setting the **Compressibility** parameter to `On`.
- On the pneumatic side, the mass flow rate and heat flow computations assume that the gas is ideal.
- In **One mechanical port** configuration, the cylinder is always assumed be grounded.
- In **Two mechanical ports** configuration, no loading on piston rod, such as inertia, friction, spring, and so on, is taken into account. If necessary, you can easily add them by connecting an appropriate building block to cylinder port R.

Parameters

- “Hydraulic Side Tab” on page 1-921
- “Pneumatic Side Tab” on page 1-922
- “Hard Stop Tab” on page 1-922

Hydraulic Side Tab

Hydraulic side piston area

Effective piston area on the hydraulic side. The default value is `2e-3` m^2.

Stroke

Piston maximum travel between caps. The default value is `0.2` m.

Piston initial distance from hydraulic port H

The distance between the piston and the cap on the hydraulic side at the beginning of simulation. This value cannot exceed the piston stroke. The default value is `0`.

Compressibility

This parameter is visible only in the **Two mechanical ports** configuration. Specifies whether fluid density on the hydraulic side is taken as constant or varying with pressure. The default value is `Off`, in which case the block models an ideal transducer. If you select `On`, the block dialog box displays additional parameters that let you model dynamic variations of the liquid density without adding any extra blocks.

Hydraulic side dead volume

This parameter is visible only when the **Compressibility** parameter is set to `On`. Fluid volume in hydraulic chamber that remains in the chamber after the rod is fully retracted. The default value is `1e-4` m^3.

Specific heat ratio

This parameter is visible only when the **Compressibility** parameter is set to `On`. Gas-specific heat ratio. The default value is `1.4`.

Initial liquid pressure (relative)

This parameter is visible only when the **Compressibility** parameter is set to `On`. The initial relative pressure of fluid in the hydraulic converter. This parameter specifies the initial condition for use in block's initial state at the beginning of a simulation run. The default value is `0`.

Converter orientation

This parameter is visible only in the **Two mechanical ports** configuration. Specifies hydraulic cylinder orientation with respect to the globally assigned positive direction. The cylinder can be installed in two different ways, depending upon whether it exerts force in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port H exerts force in negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Pneumatic Side Tab

Pneumatic side piston area

Effective piston area on the pneumatic side. The default value is `1e-3` m^2.

Pneumatic side dead volume

Gas volume in pneumatic chamber that remains in the chamber after the rod is fully retracted. The default value is `1e-4` m^3.

Initial gas pressure (absolute)

The initial absolute gas pressure in the pneumatic chamber. The default value is `101325` Pa.

Initial gas temperature

The initial gas temperature in the pneumatic chamber. The default value is `293.15` K.

Hard Stop Tab

Hard stop stiffness

The hard stop model implemented in the block assumes that the stop resists penetration of the piston with force proportional to the penetration. This parameter sets the stiffness of the contact between the stop and the piston. The default value is `1e6` N/m.

Hard stop damping

The damping in the hard stop model accounts for dissipation in the piston-stop contact. The default value is `150` N*s/m.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Compressibility**
- **Converter orientation**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the values of the **Compressibility** parameter at the time the model entered Restricted mode.

Ports

The block has the following ports:

H

Hydraulic conserving port associated with the actuator hydraulic chamber.

P

Pneumatic conserving port associated with the actuator pneumatic chamber.

E

Thermal conserving port associated with the gas in the pneumatic chamber. You can simulate the heat exchange with the environment through this port.

L

Mechanical translational conserving port associated with the actuator piston. You can model the load on the piston, such as external force, inertia, friction, or spring, and connect it through this port.

R

Mechanical translational conserving port associated with the actuator clamping structure, which is exposed by selecting the **Two mechanical ports** variant. You can model the load on the actuator casing through this port.

Introduced in R2012b

Poppet Valve

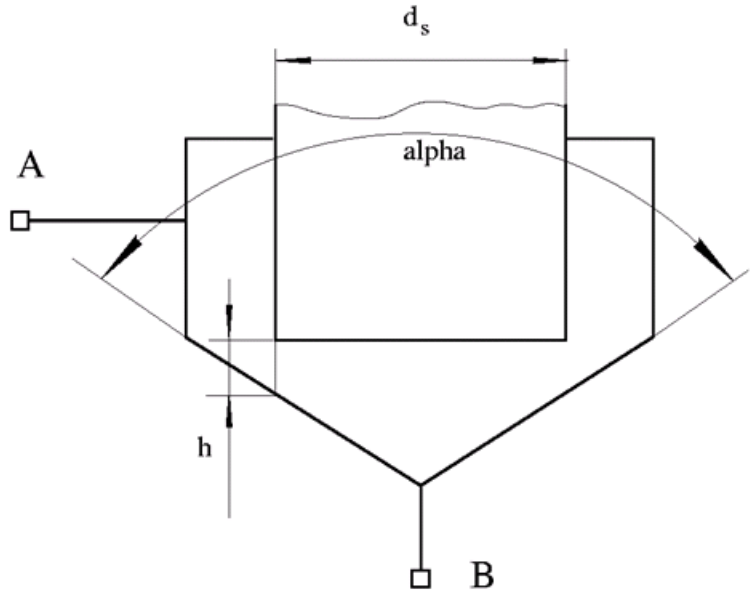
Hydraulic poppet valve

Library

Flow Control Valves

Description

The Poppet Valve block models a variable orifice created by a cylindrical sharp-edged stem and a conical seat.



The flow rate through the valve is proportional to the valve opening and to the pressure differential across the valve. The flow rate is determined according to the following equations:

$$q = C_D \cdot A(h)\sqrt{\frac{2}{\rho}}\frac{\Delta p}{\left(\Delta p^2 + p_{\mathrm{Cr}}^2\right)^{1/4}},$$

$$\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}},$$

$$h = x_0 + x$$

$$h_{\max} = d_s \frac{\sqrt{1+\cos\frac{\alpha}{2}}-1}{\sin\alpha}$$

$$A(h) = \begin{cases} A_{leak} & \text{for } h <= 0 \\ \pi \cdot \left(d_s + h \cdot \sin\frac{\alpha}{2} \cdot \cos\frac{\alpha}{2} \right) \cdot h \cdot \sin\frac{\alpha}{2} + A_{leak} & \text{for } 0 < h < h_{\max} \\ A_{\max} + A_{leak} & \text{for } h >= h_{\max} \end{cases}$$

$$A_{\max} = \frac{\pi d_s^2}{4}$$

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
x_0	Initial opening
x	Stem displacement from initial position
h	Valve opening
$h_{\max}$	Maximum valve opening. The passage area remains constant and equal to A_{max} after this.
d_s	Stem diameter
α	Cone angle
ρ	Fluid density
A_{leak}	Closed valve leakage area
$A_{\max}$	Maximum valve open area
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$

$$p_{avg} = (p_A + p_B)/2$$

where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

D_H	Valve instantaneous hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as

$\Delta p = p_A - p_B$, . Positive signal at the physical signal port S opens the valve.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- The flow passage area is assumed to be equal to the frustum side surface area.

Parameters

Valve stem diameter

The diameter of the valve stem. The default value is `0.01` m.

Seat cone angle

The angle of the valve conical seat. The parameter value must be in the range between 0 and 180 degrees. The default value is `120` degrees.

Initial opening

The initial opening of the valve. The parameter value must be nonnegative. The default value is `0`.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.65`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `10`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion

of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12 m^2`.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the valve inlet.

`B`

Hydraulic conserving port associated with the valve outlet.

`S`

Physical signal port to control spool displacement.

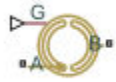
See Also

Ball Valve | Gate Valve | Needle Valve | Pressure-Compensated Flow Control Valve

Introduced in R2006a

Porting Plate Variable Orifice

Variable orifice between piston and porting plate

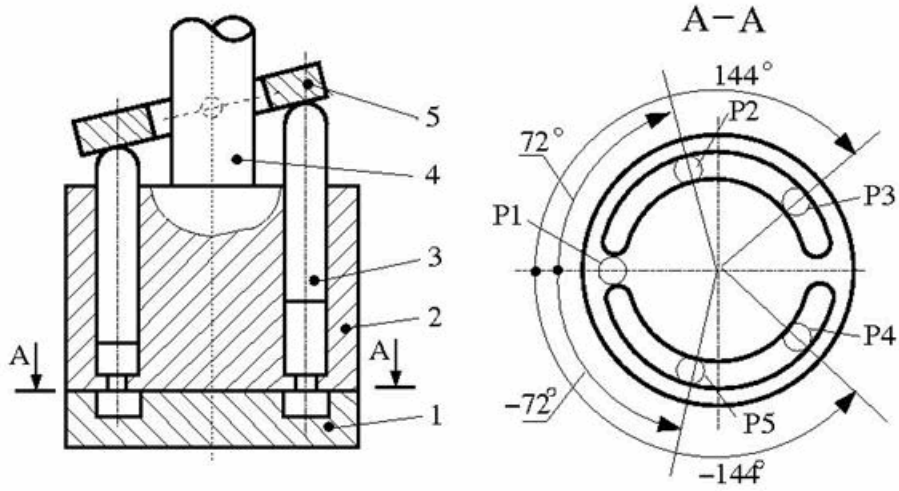


Library

Pumps and Motors

Description

Porting plate is a key element of axial-piston machines. Its objective is to provide communication between pistons and pump ports during the rotor, or cylinder block, rotation. The porting plate is equipped with two crescent-shaped slots, one of which is connected to the intake port while the other is channeled to the discharge port. The pistons are carried along the porting plate slots, thus periodically connecting the piston to either the intake or the discharge port of the machine.



The preceding diagram shows an axial-piston machine with five pistons, where:

1 Porting plate

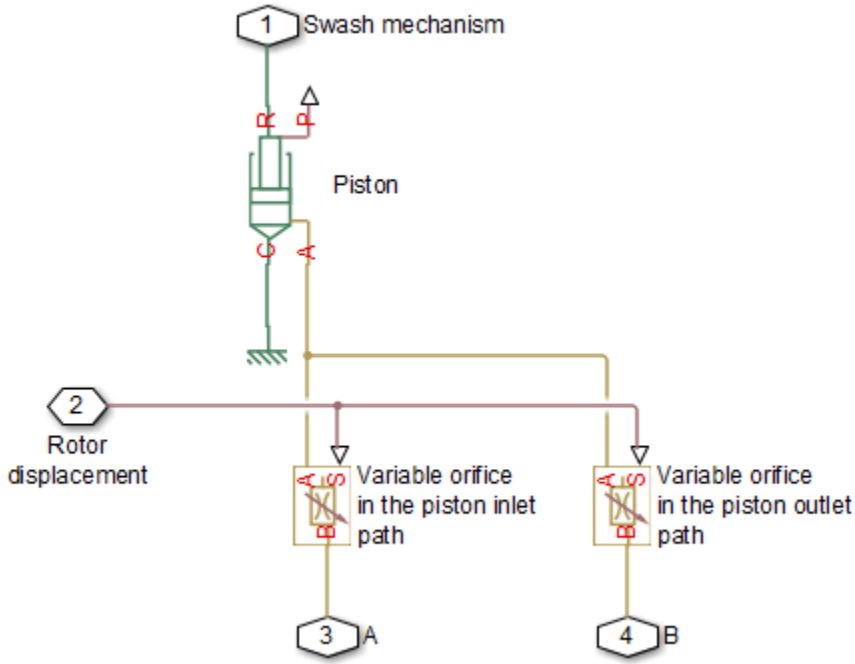
2 Rotor

3 Piston

4 Driving shaft

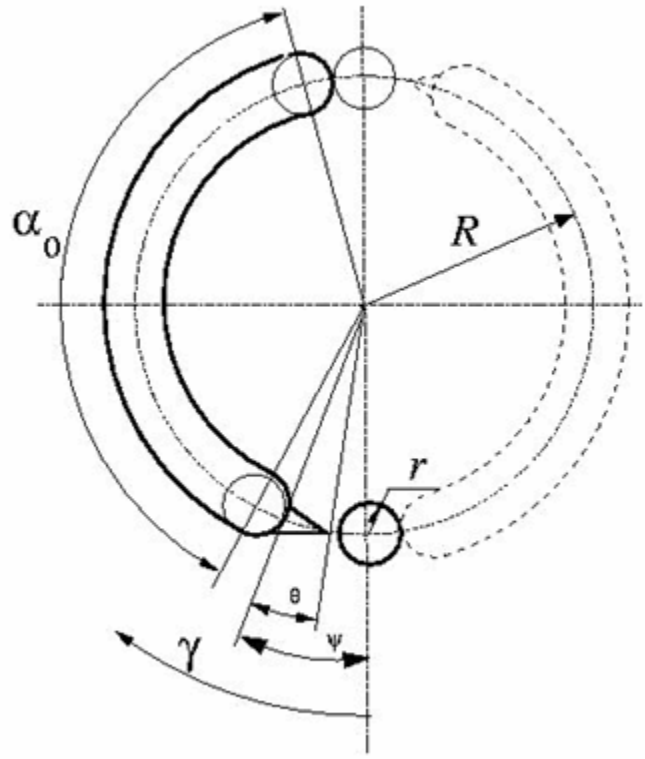
5 Swash plate

During rotor rotation, every piston gets connected to one of the porting slots. These piston connections, modeled as variable orifices, are reflected in the schematic model of a piston, shown in the following illustration.



The Porting Plate Variable Orifice block is a model of a variable orifice created between the piston chamber and a slot on a porting plate. Two Porting Plate Variable Orifice blocks are necessary in the model of a piston, to simulate connection to the intake and the discharge port, respectively.

The calculation diagram of the porting plate variable orifice is shown in the next illustration.



The model assumes that the slot is shifted by a pressure carryover angle ψ from the reference point in the direction of orifice rotation. Generally, $r / R \le \psi \le 3 \cdot r / R$, where r is the orifice radius and R is the piston pitch radius. A small triangular-shaped transition slot is placed at the beginning of the major slot, to avoid sudden pressure change. To avoid direct connection between the intake and discharge slots, the transition slot angle θ must be less than $\psi - r / R$. The transition slot area is assumed to be linearly dependent on the rotation angle and characterized by its maximum area. There are six distinctive angles defining the relationship between the rotation angle γ and the orifice opening, as listed in the following table.

No	Nomenclature	Value	Description
1	γ_1	$\psi - \theta - r / R$	Opening of transition slot starts.

No	Nomenclature	Value	Description
2	γ_2	$\psi - r / R$	Opening of the major slot starts. The orifice contacts with the transition slot.
3	γ_3	$\psi + r / R$	Major slot full opening starts.
4	γ_4	$\psi + r / R + 0.01$	Transition slot ends.
5	γ_5	$\pi - 2 \cdot r / R$	Major slot full opening ends.
6	γ_6	π	Major slot opening ends.

The variable orifice is fully opened in the α_0 region

$$\psi + r / R \le \gamma \le \pi - 2 \cdot r / R$$

where γ is the rotation angle.

The orifice area is computed with the following equations:

$$\begin{aligned}
A &= A_{leak} && \text{for } \gamma_6 < \gamma <= \gamma_1 \\
A &= G \cdot (\gamma - \gamma_1) + A_{leak} && \text{for } \gamma_1 < \gamma <= \gamma_2 \\
A &= A_{tr} + r^2 (\beta - \sin\beta) + A_{leak} && \text{for } \gamma_2 < \gamma <= \gamma_3 \\
A &= A_{tr} \cdot (1 - 100 \cdot (\gamma - \gamma_3)) + \pi \cdot r^2 + A_{leak} && \text{for } \gamma_3 < \gamma <= \gamma_4 \\
A &= \pi \cdot r^2 + A_{leak} && \text{for } \gamma_4 < \gamma <= \gamma_5 \\
A &= r^2 (\beta - \sin\beta) + A_{leak} && \text{for } \gamma_5 < \gamma <= \gamma_6
\end{aligned}$$

$$\beta = \begin{cases} 2 \cdot \alpha \cos\left(R / r \cdot \sin\left(0.5 \cdot (\gamma_3 - \gamma)\right)\right) & \text{for } \gamma_2 < \gamma <= \gamma_3 \\ 2 \cdot \alpha \cos\left(R / r \cdot \sin\left(0.5 \cdot (2 \cdot r / R + \gamma - \pi)\right)\right) & \text{for } \gamma_5 < \gamma <= \gamma_6 \end{cases}$$

where

A	Orifice area
A_{tr}	Transition slot maximum area
A_{leak}	Closed orifice leakage area

After the area has been determined, the flow rate through the orifice is computed with the following equations:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

$$\Delta p = p_A - p_B,$$

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr} \cdot \nu}{C_D \cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
D_H	Orifice hydraulic diameter
ρ	Fluid density
ν	Fluid kinematic viscosity
p_{cr}	Minimum pressure for turbulent flow
Re_{cr}	Critical Reynolds number

The Porting Plate Variable Orifice block model is essentially a building block intended for use in various piston machines. The model accounts for the flow regime by computing the Reynolds number and comparing it with its critical value. No inertial effects are considered in the model.

Connections A and B are hydraulic conserving ports associated with the inlet and outlet of the orifice. Connection G is a physical signal port associated with the input signal for the cylinder block angle. The input signal at port G is treated as an angle, in radians,

therefore it is essential that the input signal representing angular displacement is kept within the range from 0 to 2π. Connect port G to the output port of an Angle Sensor block to meet this requirement.

The flow rate is considered positive if it flows from A to B. At initial position, the orifice is assumed to be shifted by the pressure carryover angle from the slot. When the block rotates in the positive direction, the orifice starts opening. To adjust the initial position of the orifice with respect to the slot, use the **Phase angle** parameter.

Basic Assumptions and Limitations

- The model accounts for the viscous friction in the piston-plate contact.
- No inertial effects are considered.
- The plate angular displacements are considered to be small.
- The joint between the piston and the plate permanently maintains contact between the piston and the plate.

Parameters

Piston pitch radius

The radius of the pitch circle where the pistons are located. The parameter must be greater than zero. The default value is `0.05` m.

Orifice diameter

The diameter of the orifice at the bottom of the piston chamber. The parameter must be greater than zero. The default value is `0.005 m`.

Pressure carryover angle

The angle introduced between to slot and the orifice to avoid abrupt pressure change and decrease leakage. This angle is marked ψ in the preceding calculation diagram. The angle is expected to be in the range $r / R \le \psi \le 3 \cdot r / R$, where r is the orifice radius and R is the piston pitch radius. The default value is `0.06` rad.

Phase angle

This parameter sets the orifice initial angular position with respect to the slot. The default value is `0`, which means that the orifice is shifted by the pressure carryover angle from the slot.

Transition slot angle

The angle covered by a small, triangular-shaped transition slot placed before the major slot to avoid sudden pressure change, as shown in the preceding calculation diagram. To avoid direct connection between the intake and discharge slots, the transition slot angle θ must be less than $\psi - r / R$. The default value is `0.01` rad.

Transition slot maximum area

The maximum cross-sectional area of the transition slot. The slot area is assumed to be linearly-dependent on the rotation angle and reaches its maximum when the orifice approaches the major slot. The default value is `1e-6` m.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.6`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges.

Leakage area

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-9` m^2.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

G

Physical signal input port that conveys the angular position of the orifice to the block. The signal applied to the port is treated as an angle, in radians, and must be in the range between 0 and 2π.

Examples

The Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control example models a test rig designed to investigate interaction between an axial-piston pump and a typical control unit, simultaneously performing the load-sensing and pressure-limiting functions. To assure required accuracy, the model of the pump must account for such features as interaction between pistons, swash plate, and porting plate, which makes it necessary to build a detailed pump model.

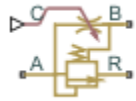
See Also

Angle Sensor | Swash Plate

Introduced in R2011a

Pressure-Compensated 3-Way Flow Control Valve

Hydraulic pressure compensating 3-way valve

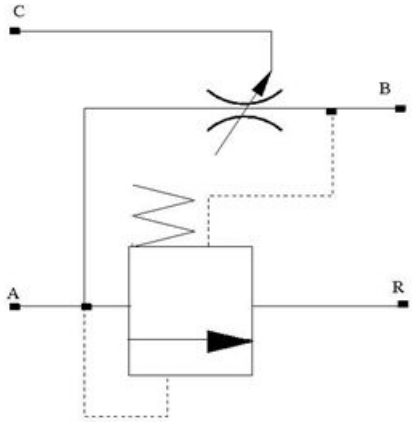


Library

Flow Control Valves

Description

The Pressure-Compensated 3-Way Flow Control Valve block represents a 3-way pressure-compensated flow control valve as a data-sheet-based model. The valve model includes a variable orifice and a normally closed pressure-control valve, connected in parallel with the orifice. The purpose of the pressure-control valve is to maintain a preset pressure differential across the orifice by diverting some flow from port A to the tank (port R) if the pressure differential exceeds the preset value. Port C controls the orifice opening, as shown in the following figure.



Depending on data listed in the manufacturer's catalogs or data sheets for your particular valve, you can choose one of the following model parameterization options:

- `By maximum area and opening` — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.
- `By area vs. opening table` — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement *A=A(h)*.

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed if the initial orifice opening is set to zero and the control member position also equals zero. The maximum orifice opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table *A=A(h)*.

The pressure-control valve representation does not account for inertia, friction, or hydraulic forces. The valve has the following area-pressure differential relationship:

$$A_{pc} = \begin{cases} A_{leak} & \text{for } p < p_{set} \\ (p - p_{set}) \dfrac{A_{\max_pc}}{p_{reg}} & \text{for } p_{set} \le p \le \left(p_{set} + p_{reg}\right) \\ A_{\max_pc} & \text{for } p > \left(p_{set} + p_{reg}\right) \end{cases}$$

where

A_{pc}	Pressure-control valve passage area
p	Pressure differential across the orifice
p_{set}	Preset pressure differential
p_{reg}	Regulation range
A_{max_pc}	Pressure-control valve maximum area
A_{leak}	Closed orifice leakage area for pressure compensator

Both for the variable orifice and the pressure compensator, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

After the area has been determined, the block computes the flow rate for both the orifice and the pressure compensator according to the following equations:

$$q = C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

$$A = \begin{cases} A_{pc} & \text{for pressure compensator} \\ A_{or} & \text{for variable orifice} \end{cases}$$

$$A_{pc} = \begin{cases} h \bullet A_{\max} / h_{\max} + A_{leak} & \text{for } h > 0 \\ A_{leak} & \text{for } h <= 0 \end{cases}$$

$$h = x_0 + x$$

$$p = \begin{cases} p_A - p_R & \text{for pressure compensator} \\ p_A - p_B & \text{for variable orifice} \end{cases}$$

$$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

$$D_H = \sqrt{\frac{4A}{\pi}}$$

where

q	Flow rate
p	Pressure differential
p_A, p_B, p_R	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
A	Instantaneous orifice passage area
A_{max}	Orifice maximum area
A_{leak}	Closed orifice leakage area
h_{max}	Control member maximum displacement
x_0	Initial opening
x	Control member displacement from initial position
h	Orifice opening
ρ	Fluid density
ν	Fluid kinematic viscosity
p_{cr}	Minimum pressure for turbulent flow
Re_{cr}	Critical Reynolds number
D_H	Instantaneous orifice hydraulic diameter

Connections A, B, and R are the conserving hydraulic ports associated with the valve inlet, outlet, and return, respectively. Connection C is a physical signal port through which the orifice opening is controlled. The block positive direction is from port A to port B. Positive signal at port C opens the valve.

Assumptions and Limitations

Fluid inertia and pressure compensator loading are not taken into account.

Parameters

- "Variable Orifice Tab" on page 1-943
- "Pressure Compensator Tab" on page 1-946

Variable Orifice Tab

Orifice parameterization

Select one of the following methods for specifying the orifice:

- `By maximum opening and area` — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By orifice area vs. orifice opening table` — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Orifice maximum area

The area of a fully opened orifice. The parameter value must be greater than zero. The default value is `1e-4` m^2. This parameter is used if **Orifice parameterization** is set to `By maximum opening and area`.

Orifice maximum opening

The maximum displacement of the control member. The parameter value must be greater than zero. The default value is `0.005` m. This parameter is used if **Orifice parameterization** is set to `By maximum opening and area`.

Tabulated orifice openings

Vector of input values for orifice openings, specified as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you

must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in mm, are `[-6, 0, 1, 3, 6, 9, 15]`. This parameter is used if **Orifice parameterization** is set to `By orifice area vs. orifice opening table`. **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup.

Tabulated orifice area

Vector of orifice areas, specified as a one-dimensional array. The vector must be the same size as the orifice openings vector. All the values must be positive. The default values, in cm^2, are `[1e-7, 1e-6, 0.08, 0.145, 0.96, 1, 1.001]`. This parameter is used if **Orifice parameterization** is set to `By orifice area vs. orifice opening table`.

Interpolation method

This parameter is used if **Orifice parameterization** is set to `By orifice area vs. orifice opening table`. Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page.

Extrapolation method

This parameter is used if **Orifice parameterization** is set to `By orifice area vs. orifice opening table`. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page.

Orifice initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or 0 for zero-lap configuration. The default value is `0`.

Orifice flow discharge coefficient

Semi-empirical coefficient that is used in the computation of flow rate through the orifice. Its value depends on the geometrical properties of the orifice and usually is provided in textbooks or manufacturer data sheets. The default value is `0.6`.

Orifice laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Orifice laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Orifice critical Reynolds number** parameter.

Orifice laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Orifice laminar transition specification** parameter is set to `Pressure ratio`.

Orifice critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Orifice laminar transition specification** parameter is set to `Reynolds number`.

Orifice leakage area

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-9` m^2.

Pressure Compensator Tab

Pressure differential across orifice setting

Pressure difference that must be maintained across the variable orifice by the pressure compensator. The default value is `6e5` Pa.

Pressure differential regulation range

Pressure increase over the preset level needed to fully open the pressure compensator orifice. The lower the value of the range, the higher the valve sensitivity. The default value is `0.5e5` Pa.

Pressure compensator maximum area

The area of a fully opened pressure compensator. The parameter value must be greater than zero. The default value is `1e-4` m^2.

Pressure compensator flow discharge coefficient

Semi-empirical coefficient that is used in the computation of flow rate through the pressure compensator. The default value is `0.6`.

Pressure compensator laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Pressure compensator laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Pressure compensator critical Reynolds number** parameter.

Pressure compensator laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Pressure compensator laminar transition specification** parameter is set to `Pressure ratio`.

Pressure compensator critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Pressure compensator laminar transition specification** parameter is set to `Reynolds number`.

Pressure compensator leakage area

The total area of possible leaks in the completely closed orifice. Must be greater than 0. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The default value is `1e-9` m^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**
- **Extrapolation method**
- **Orifice laminar transition specification**
- **Pressure compensator laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the valve inlet.

B

Hydraulic conserving port associated with the valve outlet.

R

Hydraulic conserving port that connects with the tank.

C

Physical signal control port.

See Also

Pressure-Compensated Flow Control Valve

Introduced in R2013b

Pressure-Compensated Flow Control Valve

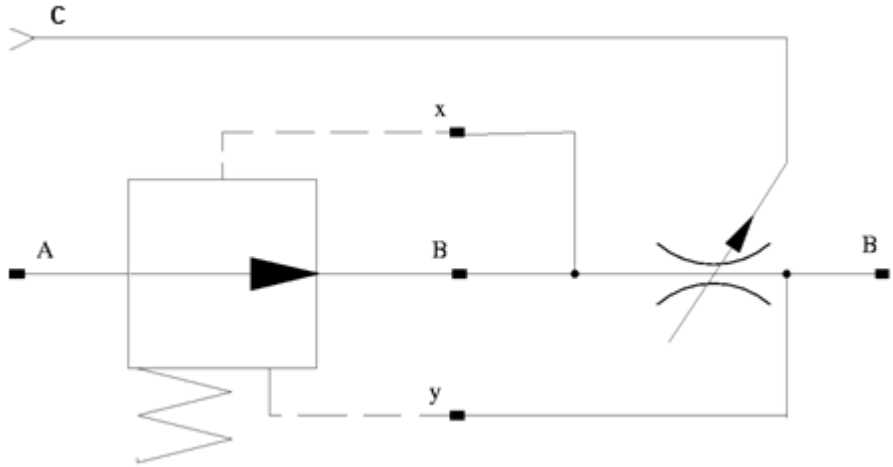
Hydraulic pressure compensating valve

Library

Flow Control Valves

Description

The Pressure-Compensated Flow Control Valve block represents a pressure-compensated flow control valve as a data-sheet-based model. The valve is based on a Pressure Compensator block installed upstream from a Variable Orifice block, as shown in the following illustration.



Depending on data listed in the manufacturer's catalogs or data sheets for your particular valve, you can choose one of the following model parameterization options:

- `By maximum area and opening` — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.
- `By area vs. opening table` — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement *A=A(h)*.

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table *A=A(h)*. In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure differential is determined as

$\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}}$, . Positive signal at port C opens the valve.

Assumptions and Limitations

Fluid inertia is not taken into account.

Parameters

Model parameterization

Select one of the following methods for specifying the orifice:

- `By maximum area and opening` — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.

- `By area vs. opening table` — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Orifice maximum area

Specify the area of a fully opened orifice. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Orifice maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-4` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Tabulated orifice openings

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-2e-3,0,5e-3,15e-3]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`. **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup.

Tabulated orifice area

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in m^2, are `[1e-12,4e-12,1.e-5,1.02e-5]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Interpolation method

This parameter is used if **Model parameterization** is set to `By area vs. opening table`. Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page.

Extrapolation method

This parameter is used if **Model parameterization** is set to `By area vs. opening table`. Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page.

Pressure differential across the orifice

Pressure difference that must be maintained across the element by the pressure compensator. The default value is `6e5` Pa.

Pressure reducing valve regulation range

Pressure increase over the preset level needed to fully close the valve. Must be less than 0.2 of the **Pressure differential across the orifice** parameter value. The default value is `5e4` Pa.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is `0`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**
- **Extrapolation method**
- **Laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the valve inlet.

`B`

Hydraulic conserving port associated with the valve outlet.

`C`

Physical signal control port.

See Also

Ball Valve | Gate Valve | Needle Valve | Poppet Valve

Introduced in R2006a

Pressure Compensator

Valve used to regulate the pressure drop across a hydraulic component

Library: Simscape / Fluids / Hydraulics (Isothermal) / Valves / Pressure Control Valves

Description

The Pressure Compensator block models the flow through a valve that constricts so as to maintain a preset pressure drop between a chosen two hydraulic nodes. The valve has four hydraulic ports, two being flow passages (the inlet, **A**, and the outlet, **B**) and two pressure sensors (**X** and **Y**). The normally open valve contracts when the pressure drop from **X** to **Y** rises above the valve pressure setting. The drop in opening area is a function of the pressure drop—proportional to it in a linear parameterization (the block default) or a general function of it in a tabulated parameterization. The valve serves its purpose until it hits the limit of its pressure regulation range—a point at which the valve is fully closed and the pressure drop can again rise unabated.

Valve Opening

The opening area calculation depends on the valve parameterization selected for the block: either `Linear area-opening relationship` or `Tabulated data - Area vs. pressure`.

If the **Valve parameterization** block parameter is in the default setting of `Linear area-opening relationship`, the opening area is computed as:

$$S(\Delta p_{xy}) = S_{Max} - k\left(\Delta p_{xy} - \Delta p_{Set}\right),$$

where:

- S_{Max} is the value specified in the **Maximum passage area** block parameter.
- Δp_{Set} is the value specified in the **Valve pressure setting** block parameter.

- Δp_{XY} is the pressure drop from port **X** to port **Y**:

  $$\Delta p_{XY} = p_X - p_Y,$$

  where p is the gauge pressure at the port indicated by the subscript (**X** or **Y**).
- k is the linear constant of proportionality:

  $$k = \frac{S_{Max} - S_{Leak}}{\Delta p_{Reg}},$$

  where in turn:

  - S_{Leak} is the value specified in the **Leakage area** block parameter.
  - Δp_{Reg} is that specified in the **Valve regulation range** block parameter.

At and below the valve pressure setting, the opening area is that of a fully open valve:

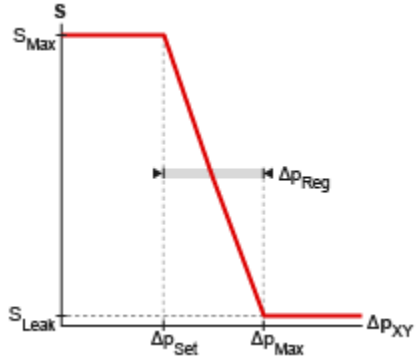
$$S(\Delta p_{XY} \le \Delta p_{Set}) = S_{Max}.$$

At and above a maximum pressure, the opening area is that due to internal leakage alone:

$$S(\Delta p \ge \Delta p_{Max}) = S_{Leak},$$

where the maximum pressure drop Δp_{Max} is the sum:

$$\Delta p_{Max} = \Delta p_{Set} + \Delta p_{Reg}.$$

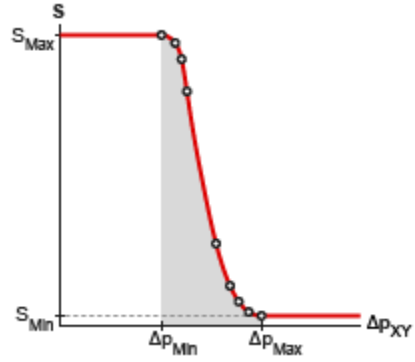


Opening area in the `Linear area-opening relationship` parameterization

If the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. pressure`, the opening area is computed as:

$$S = S(\Delta p_{XY}),$$

where S_{XY} is a tabulated function constructed from the **Pressure drop vector** and **Opening area vector** block parameters. The function is based on linear interpolation (for points within the data range) and nearest-neighbor extrapolation (for points outside the data range). The leakage and maximum opening areas are the minimum and maximum values of the **Valve opening area vector** block parameter.



Opening area in the `Tabulated data - Area vs. pressure` parameterization

By default, the valve opening dynamics are ignored. The valve is assumed to respond instantaneously to changes in the pressure drop, without time lag between the onset of a pressure disturbance and the increased valve opening that the disturbance produces. If such time lags are of consequence in a model, you can capture them by setting the **Opening dynamics** block parameter to `Include valve opening dynamics`. The valves then open each at a rate given by the expression:

$$\dot{S} = \frac{S(\Delta p_{SS}) - S(\Delta p_{In})}{\tau},$$

where τ is a measure of the time needed for the instantaneous opening area (subscript `In`) to reach a new steady-state value (subscript `SS`).

The primary purpose of the leakage area of a closed valve is to ensure that at no time does a portion of the hydraulic network become isolated from the remainder of the model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage is generally present in minuscule amounts in real valves but in a model its exact value is less important than it being a small number greater than zero. The leakage area is obtained from the block parameter of the same name.

Valve Flow Rate

The causes of the pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is captured in the block by the discharge coefficient, a measure of the flow rate through the valve relative to the theoretical value that it would have in an ideal valve. The flow rate through the valve is defined as:

$$q = C_D S \sqrt{\frac{2}{\rho}} \frac{\Delta p_{AB}}{\left[(\Delta p_{AB})^2 + p_{Crit}^2 \right]^{1/4}},$$

where:

- q is the volumetric flow rate through the valve.

- C_D is the value of the **Discharge coefficient** block parameter.
- S is the opening area of the valve.
- Δp_{AB} is the pressure drop from port **A** to port **B**.
- p_{Crit} is the pressure differential at which the flow shifts between the laminar and turbulent flow regimes.

The calculation of the critical pressure depends on the setting of the **Laminar transition specification** block parameter. If this parameter is in the default setting of `By pressure ratio`:

$$p_{Crit} = \left(p_{Atm} + p_{Avg} \right) \left(1 - \beta_{Crit} \right),$$

where:

- p_{Atm} is the atmospheric pressure (as defined for the corresponding hydraulic network).
- p_{Avg} is the average of the gauge pressures at ports **A** and **B**.
- β_{Crit} is the value of the **Laminar flow pressure ratio** block parameter.

If the **Laminar transition specification** block parameter is instead set to `By Reynolds number`:

$$p_{Crit} = \frac{\rho}{2} \left(\frac{\mathrm{Re}_{Crit} \nu}{C_D D_H} \right)^2,$$

where:

- Re_{Crit} is the value of the **Critical Reynolds number** block parameter.
- ν is the kinematic viscosity specified for the hydraulic network.
- D_H is the instantaneous hydraulic diameter:

$$D_H = \sqrt{\frac{4S}{\pi}}.$$

Ports

Conserving

A — Valve inlet
hydraulic (isothermal liquid)

Opening through which the flow can enter the valve.

B — Valve outlet
hydraulic (isothermal liquid)

Opening through which the flow can exit the valve.

X — Pressure sensor
hydraulic (isothermal liquid)

Port at which pressure is measured for the purpose of setting the valve opening area.

Y — Pressure sensor
hydraulic (isothermal liquid)

Port at which pressure is measured for the purpose of setting the valve opening area.

Parameters

`Opening area parameterization` — Method by which to calculate the opening area of the valve
`Linear area-opening relationship` (default) | `Tabulated data - Area vs. pressure`

Method by which to calculate the opening area of the valve. The default setting prescribes a linear relationship between the opening area of the valve and the pressure drop between its sensor ports. The alternative setting allows for a general, nonlinear relationship to be specified in tabulated form.

`Maximum passage area` — Opening area of the valve in the fully open position
`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position. The valve is fully open if the pressure drop from port **X** to port **Y** is less than the value specified in the **Valve pressure setting** block parameter.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve pressure setting` — Pressure drop to maintain from port X to port Y

`30e5 Pa` (default) | positive scalar in units of pressure

Pressure drop from port **X** to port **Y** that the valve is to maintain. The valve closes in proportion to the pressure drop if it should exceed the value specified here. If the pressure drop is less than this value, the opening area is that specified in the **Maximum passage area** block parameter.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve regulation range` — Pressure drop interval over which the valve is to designed to operate

`1.5e5 Pa` (default) | positive scalar in units of pressure

Pressure drop interval over which the valve is designed to operate. This interval stretches from the largest pressure drop at which the valve is fully open to the lowest pressure drop at which the valve is fully closed. Only within this range is the valve capable of maintaining the specified pressure setting.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates

`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

Leakage area — Opening area of the valve in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

Pressure differential vector — Vector of pressure drops at which to tabulate the valve opening areas

`[30.15E5 : 15E3 : 31.35E5] Pa` (default) | vector of positive numbers in units of pressure

Vector of pressure differentials from port **X** to port **Y** at which to specify the opening area of the valve. The vector elements must increase monotonically from left to right. This order is important when specifying the **Opening area vector** block parameter.

The block uses this data to construct a lookup table by which to determine from the pressure differential the valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

Opening area vector — Vector of opening areas at the specified pressure differential breakpoints

`[9E-5 : -1 : 1E-5] m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Pressure differential vector** block parameter. The vector elements must decrease monotonically from left to right (with increasing pressure). For best results, avoid regions of flattened slope.

The block uses this data to construct a lookup table by which to determine from the pressure differential the valve opening area. Data is handled with linear interpolation

(within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Laminar transition specification` — Parameter in terms of which to define the boundary between laminar and turbulent regimes

`Pressure ratio` (default) | `Reynolds number`

Parameter in terms of which to specify the boundary between the laminar and turbulent flow regimes. The pressure ratio of the default parameterization is defined as the gauge pressure at the outlet divided by the same at the inlet.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow is assumed to transition between laminar and turbulent regimes. The pressure ratio is defined as the gauge pressure at the outlet divided by the same at the inlet. The transition is assumed to be smooth and centered on this value.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Reynolds number`.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes

12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Reynolds number`.

`Opening dynamics` — Choice of whether to capture the opening dynamics of the valve

`Do not include valve opening dynamics` (default) | `Include valve opening dynamics`

Choice of whether to capture the opening dynamics of the valve. Selecting `Include valve opening dynamics` causes the valve to open gradually, so as to approach its new steady-state area over a small time span. The characteristic time for such transitions is given in the **Opening time constant** block parameter.

Setting this parameter to the alternative `Do not include valve opening dynamics` is equivalent to specifying a value of `0` for the opening time constant. The opening area is in this case assumed to reach its new steady-state value instantaneously.

Include opening dynamics to more accurately capture the behavior of a real valve. For best real-time simulation performance, use with a local solver or disable valve opening dynamics altogether.

`Opening time constant` — Characteristic time scale of the valve opening transitions

`0.1 s` (default) | positive scalar in units of time

Measure of the time taken by the valve to transition from its current opening area to a new steady-state value. The block uses this parameter to calculate the rate at which a valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

`Initial area` — Opening area at the start of simulation

`1e-12 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow within the valve at the start of simulation. The block uses this parameter to calculate the initial rate at which the valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

See Also

Pressure Reducing 3-Way Valve | Pressure Reducing Valve | Pressure Relief Valve

Introduced in R2016a

Pressure Reducing 3-Way Valve

Composite valve comprising the functions of pressure reducing and relief valves

Library: Simscape / Fluids / Hydraulics (Isothermal) / Valves / Pressure Control Valves

Description

The Pressure Reducing 3-Way Valve block models the flow through a valve that constricts and, if necessary, vents (via a separate line) so as to maintain a preset pressure differential between its outlet (port **A**) and its surroundings (**T**, the vent). The valve combines the functions of two more elementary valves, one a pressure-reducing valve, the other a pressure-relief valve.

The pressure-reducing valve runs between the inlet (**P**) and the outlet (**A**). It is normally open but closes as needed to combat pressure fluctuations upstream of the outlet. The pressure-relief valve runs between the outlet (**A**) and the vent (**T**). It is normally closed but opens if the outlet pressure should exceed (by a specified transition amount) the pressure regulation range of the pressure-reducing valve.

The valve opening areas are functions of the pressure differential from port **A** to port **T**—either directly proportional to it or general (tabulated) functions of it. The opening areas can each vary up to a point—the limit of the respective valve regulation range—beyond which the valve is saturated at full capacity and unable to counteract any additional pressure rises.

Valve Opening

The opening area calculation depends on the valve parameterization selected for the block: either `Linear area-opening relationship` or `Tabulated data - Area vs. pressure`.

If the **Valve parameterization** block parameter is in the default setting of `Linear area-opening relationship`, the opening area of the pressure-reducing valve (**P**-**A**) is computed as:

$$S_{PA} = \frac{S_{Max} + S_{Leak}}{2} - \frac{S_{Max} - S_{Leak}}{2} \tanh\left[\frac{\lambda\left(\Delta p_{AT} - \Delta p_{Rdn}\right)}{\Delta p_{Reg} / 2}\right],$$

where the `tanh` term serves to smooth the transitions to the fully open and fully closed valve positions. In the equation:

- S_{Max} is the value specified in the **Maximum passage area** block parameter.
- S_{Leak} is the value specified in the **Leakage area** block parameter.
- λ is the value of the **Valve opening adjustment coefficient** block parameter, a measure of the smoothing to apply to the valve transitions. The smaller the value, the smoother the transition.
- Δp_{Reg} is the value specified in the **Valve regulation range** block parameter.
- Δp_{Rdn} is the midpoint of the pressure regulation range of the pressure-reducing valve:

  $$\Delta p_{Rdn} = \Delta p_{Set} + \frac{\Delta p_{Reg}}{2},$$

  with Δp_{Set} being the value of the **Reducing valve pressure setting** block parameter.

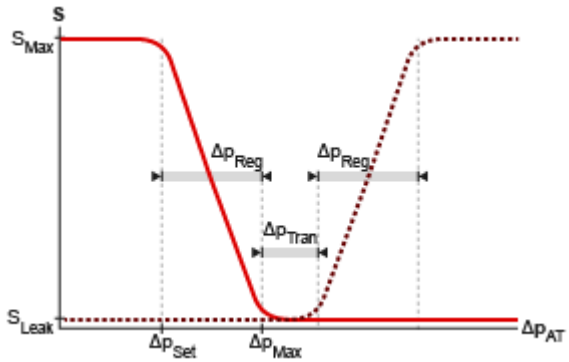
The opening area of the opening area of the pressure-relief valve (**A**-**T**) is likewise computed as:

$$S_{AT} = \frac{S_{Max} + S_{Leak}}{2} + \frac{S_{Max} - S_{Leak}}{2} \tanh\left(\frac{\lambda\left(\Delta p_{AT} - \Delta p_{Rlf}\right)}{\Delta p_{Reg} / 2}\right),$$

where Δp_{Rlf} is the midpoint of the pressure regulation range of the pressure-relief valve:

$$\Delta p_{Rlf} = \Delta p_{Set} + \Delta p_{Reg} + \Delta p_{Tran} + \frac{\Delta p_{Reg}}{2},$$

with Δp_{Tran} being the transition pressure differential—that required for the pressure-relief valve to open after the pressure-reducing valve has fully closed. This value is obtained from the **Transition pressure** block parameter.

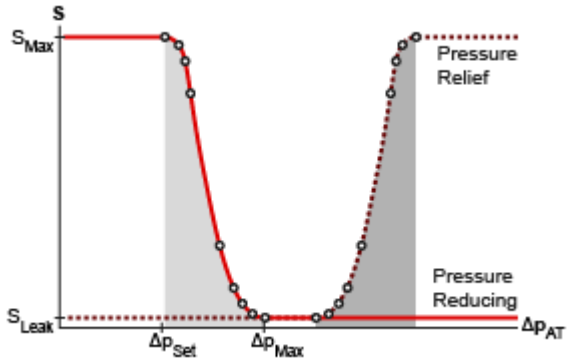


Opening area in the `Linear area-opening relationship` parameterization

If the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. pressure`, the valve opening areas are computed as:

$$S = S(\Delta p_{\mathrm{AT}}),$$

where S_{AT} is a tabulated function constructed from the **Pressure drop vector** and **Opening area vector** block parameters. The function is based on linear interpolation (for points within the data range) and nearest-neighbor extrapolation (for points outside the data range). The leakage and maximum opening areas are the minimum and maximum values of the **Valve opening area vector** block parameter.



Opening area in the `Tabulated data - Area vs. pressure` parameterization

By default, the valve opening dynamics are ignored. The valves are each assumed to respond instantaneously to changes in the pressure drop, without time lag between the onset of a pressure disturbance and the increased valve opening that the disturbance produces. If such time lags are of consequence in a model, you can capture them by setting the **Opening dynamics** block parameter to `Include valve opening dynamics`. The valves then open each at a rate given by the expression:

$$\dot{S} = \frac{S(\Delta p_{SS}) - S(\Delta p_{In})}{\tau},$$

where τ is a measure of the time needed for the instantaneous opening area (subscript `In`) to reach a new steady-state value (subscript `SS`).

The primary purpose of the leakage area of a closed valve is to ensure that at no time does a portion of the hydraulic network become isolated from the remainder of the model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage is generally present in minuscule amounts in real valves but in a model its exact value is less important than it being a small number greater than zero. The leakage area is obtained from the block parameter of the same name.

Valve Flow Rates

The causes of the pressure losses incurred in the passages of the composite valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is captured in the block by the discharge coefficient, a measure of the flow rate through a valve relative to the theoretical value that it would have in the ideal case. The flow rates through each of the elementary valves are defined as:

$$q = C_D S \sqrt{\frac{2}{\rho}} \frac{\Delta p}{\left[(\Delta p)^2 + p_{Crit}^2 \right]^{1/4}},$$

where:

- q is the volumetric flow rate through the valve (**P**-**A** or **A**-**T**)..
- C_D is the value of the **Discharge coefficient** block parameter.
- S is the opening area of the valve (S_{PA} or S_{AT}).
- Δp is the pressure drop across the valve (Δp_{PA} or Δp_{AT}).
- p_{Crit} is the pressure differential at which the flow shifts between the laminar and turbulent flow regimes.

The calculation of the critical pressure depends on the setting of the **Laminar transition specification** block parameter. If this parameter is in the default setting of `By pressure ratio`:

$$p_{Crit} = \left(p_{Atm} + p_{Avg} \right) \left(1 - \beta_{Crit} \right),$$

where:

- p_{Atm} is the atmospheric pressure (as defined for the corresponding hydraulic network).
- p_{Avg} is the average of the gauge pressures at the ports (**P** and **A** or **A** and **T**).
- β_{Crit} is the value of the **Laminar flow pressure ratio** block parameter.

If the **Laminar transition specification** block parameter is instead set to `By Reynolds number`:

$$p_{\mathrm{Crit}} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{\mathrm{Crit}}\nu}{C_{\mathrm{D}}D_{\mathrm{H}}}\right)^2,$$

where:

- Re_{Crit} is the value of the **Critical Reynolds number** block parameter.
- ν is the kinematic viscosity specified for the hydraulic network.
- D_{H} is the instantaneous hydraulic diameter:

$$D_{\mathrm{H}} = \sqrt{\frac{4S}{\pi}}.$$

Ports

Conserving

A — Valve outlet

hydraulic (isothermal liquid)

Opening through which the flow can exit the valve.

P — Valve inlet

hydraulic (isothermal liquid)

Opening through which the flow can enter the valve.

T — Valve vent

hydraulic (isothermal liquid)

Opening through which the flow can vent from the valve.

Parameters

`Opening area parameterization` — Method by which to calculate the opening area of the valve

`Linear area-opening relationship` (default) | `Tabulated data - Area vs. pressure`

Method by which to calculate the opening area of the valve. The default setting prescribes a linear relationship between the opening area of the valve and the pressure drop between the outlet and the vent opening. The alternative setting allows for a general, nonlinear relationship to be specified in tabulated form.

`Maximum passage area` — Opening areas of the P-A and A-T valves in the fully open position

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of each valve when fully open. The pressure-reducing and pressure-relief valves are assumed to be identical in size. See the block description for the pressure conditions under which the valves can each be said to be fully open.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Reducing valve pressure setting` — Pressure from port A to port T at which the reducing valve begins to close

`6e5 Pa` (default) | positive scalar in units of pressure

Pressure differential from port **A** to port **T** that the composite valve is to maintain. The pressure-reducing valve closes in proportion to the pressure differential if its value should exceed that specified here. The pressure-relief valve opens in proportion to the same if its value should exceed, by an amount given by the **Transition pressure** block parameter, that needed to fully close the reducing valve.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve regulation range` — Pressure drop interval over which the valves are each designed to operate

`1.5e5 Pa` (default) | positive scalar in units of pressure

Pressure differential interval over which the valves are each designed to operate. This interval spans those pressures over which the opening area changes between its minimum and maximum values. The value specified here applies to both the pressure-reducing and pressure-relief valves.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Transition pressure` — Pressure interval between the final closing of the reducing valve and the first opening of the relief valve

`2e5 Pa` (default) | positive scalar in units of pressure

Pressure differential in excess of that required to fully close the pressure-reducing valve at which the pressure-relief valve first opens. The greater the transition pressure, the greater the lag between the closing of the pressure-reducing valve and the opening of its pressure-relief counterpart.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve flow discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates

`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter. The value specified here is shared by both pressure-reducing and pressure-relief valves.

`Valve leakage area` — Opening area of the valve in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of a valve in the fully closed position, when only internal leakage between its respective ports remains. The leakage area is the same in the pressure-reducing and pressure-relief valves. This parameter serves primarily to ensure that closure of the valve does not cause portions of the hydraulic network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve opening adjustmen coefficient` — Smoothing factor for easing the transition to the fully open or fully closed position

`1` (default) | positive unitless scalar

Smoothing factor for easing the transition to the fully open or fully closed valve position. The smoothing enhances the numerical robustness of the model, helping to avoid numerical errors due to discontinuities and simulation slowdowns due to zero crossings. Use this parameter to scale the widths of the smoothed regions. The smaller the value, the broader the transition.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Pressure differential vector for reducing valve` — Vector of pressure drops at which to tabulate the valve opening areas

`[6.06E5 : .06E5 : 6.24E5] Pa` (default) | vector of positive numbers in units of pressure

Vector of pressure drops from port **P** to port **A** at which to specify the opening area of the valve. The vector elements must increase monotonically from left to right. This order is important when specifying the **Opening area vector for reducing valve** block parameter.

The block uses this data to construct a lookup table by which to determine from the pressure differential the reducing-valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Opening area vector for reducing valve` — Vector of opening areas at the specified pressure differential breakpoints

`[.9168, .69, .31, .0832] E-4 m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Pressure differential vector for reducing valve** block parameter. The vector elements must decrease monotonically from left to right (with increasing pressure). For best results, avoid regions of flattened slope.

The block uses this data to construct a lookup table by which to determine from the pressure differential the reducing-valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Pressure differential vector for relief valve` — Vector of pressure drops at which to tabulate the valve opening areas

`[6.2, 6.26, 6.32, 6.38] E5 Pa` (default) | vector of positive numbers in units of pressure

Vector of pressure drops from port **A** to port **T** at which to specify the opening area of the valve. The vector elements must increase monotonically from left to right. This order is important when specifying the **Opening area vector for relief valve** block parameter.

The block uses this data to construct a lookup table by which to determine from the pressure differential the reducing-valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Opening area vector for relief valve` — Vector of opening areas at the specified pressure differential breakpoints

`[.9168, .69, .31, .0832] E-4 m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Pressure differential vector for relief valve** block parameter. The vector elements must decrease monotonically from left to right (with increasing pressure). For best results, avoid regions of flattened slope.

The block uses this data to construct a lookup table by which to determine from the pressure differential the reducing-valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Laminar transition specification` — Parameter in terms of which to define the boundary between laminar and turbulent regimes

`Pressure ratio` (default) | `Reynolds number`

Parameter in terms of which to specify the boundary between the laminar and turbulent flow regimes. The pressure ratio of the default parameterization is defined as the gauge pressure at the outlet divided by the same at the inlet.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow is assumed to transition between laminar and turbulent regimes. The pressure ratio is defined as the gauge pressure at the outlet divided by the same at the inlet. The transition is assumed to be smooth and centered on this value.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Pressure ratio`.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes

12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Reynolds number`.

`Opening dynamics` — Choice of whether to capture the opening dynamics of the valve

`Do not include valve opening dynamics` (default) | `Include valve opening dynamics`

Choice of whether to capture the opening dynamics of the valve. Selecting `Include valve opening dynamics` causes the valve to open gradually, so as to approach its new steady-state area over a small time span. The characteristic time for such transitions is given in the **Opening time constant** block parameter.

Setting this parameter to the alternative `Do not include valve opening dynamics` is equivalent to specifying a value of `0` for the opening time constant. The opening area is in this case assumed to reach its new steady-state value instantaneously.

Include opening dynamics to more accurately capture the behavior of a real valve. For best real-time simulation performance, use with a local solver or disable valve opening dynamics altogether.

`Opening time constant` — Characteristic time scale of the valve opening transitions

`0.1 s` (default) | positive scalar in units of time

Measure of the time taken by the valve to transition from its current opening area to a new steady-state value. The block uses this parameter to calculate the rate at which a valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

`Initial reducing valve area` — Opening area of the reducing valve at the start of simulation

`1e-9 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow within the pressure-reducing valve at the start of simulation. The block uses this parameter to calculate the initial rate at which the

pressure-reducing valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

`Initial relief valve area` — Opening area of the relief valve at the start of simulation

`1e-9 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow within the pressure-relief valve at the start of simulation. The block uses this parameter to calculate the initial rate at which the pressure-relief valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

See Also

Pressure Compensator | Pressure Reducing Valve | Pressure Relief Valve

Introduced in R2016a

Pressure Reducing Valve

Valve used to reduce pressure to a specified value

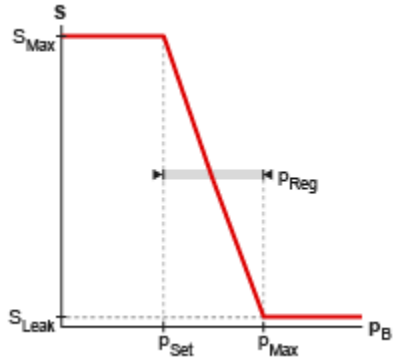
Library: Simscape / Fluids / Hydraulics (Isothermal) / Valves / Pressure Control Valves

Description

The Pressure Reducing Valve block models the flow through a valve that constricts so as to maintain a preset gauge pressure at its outlet (port **B**). The normally open valve contracts when the outlet pressure, measured against atmospheric pressure, exceeds the valve pressure setting. The drop in opening area is a function of the outlet pressure—proportional to it in a linear parameterization (the block default) or a general function of it in a tabulated parameterization. The valve serves its purpose until it hits the limit of its pressure regulation range—a point at which the valve is fully closed and the outlet pressure can again rise unabated.

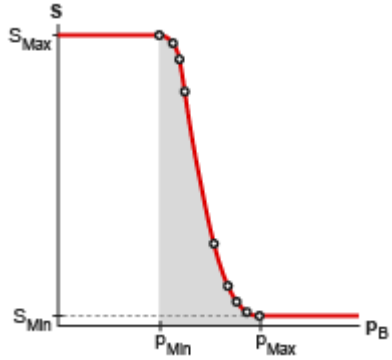
Valve Opening

The figures show the opening characteristics of the valve. In the default valve parameterization, the opening area (S) drops linearly with the outlet pressure (p_B) with its value ranging from a maximum in the fully open position (S_{Max}) to a minimum in the fully closed position (S_{Leak}). The drop occurs over a pressure range stretching from the valve pressure setting (p_{Set}) to a maximum equal to the sum of the valve regulation range (p_{Reg}) and the valve pressure setting.



Opening area in the `Linear area-opening relationship` parameterization

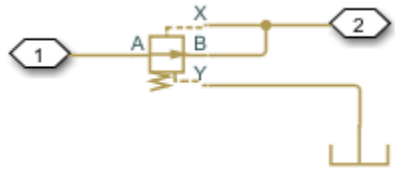
In the optional tabulated parameterization, the opening area is a general function of the outlet pressure, with its value dropping from a maximum (S_{Max} to a minimum (S_{Min}) contained in the tabulated data. The maximum opening occurs at the minimum pressure (p_{Min}) contained in the data and the minimum opening at the maximum pressure (p_{Max}). Calculations are based on linear interpolation (between the tabulated bounds) and nearest-neighbor extrapolation (outside of those bounds).



Opening area in the `Tabulated data - Area vs. pressure` parameterization

Composite Diagram

This block is a composite component with a Pressure Compensator block connected as follows: its sensor port **X** connected to its outlet (port **B**) and its sensor port **Y** connected to a hydraulic reference. The hydraulic reference is by definition at atmospheric pressure as defined for the entire hydraulic network. For detailed block calculations see the Pressure Compensator block (noting that quantities at port **X** are defined at port **B** and those at port **Y** are defined at reference conditions).



Composite model of the pressure relief valvet

Ports

Conserving

A — Valve inlet

hydraulic (isothermal liquid)

Opening through which the flow can enter the valve.

B — Valve outlet

hydraulic (isothermal liquid)

Opening through which the flow can exit the valve.

Parameters

`Opening area parameterization` — Method by which to calculate the opening area of the valve

`Linear area-opening relationship` (default) | `Tabulated data - Area vs. pressure`

Method by which to calculate the opening area of the valve. The default setting prescribes a linear relationship between the opening area of the valve and the pressure drop between its sensor ports. The alternative setting allows for a general, nonlinear relationship to be specified in tabulated form.

`Maximum passage area` — Opening area of the valve in the fully open position

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position. The valve is fully open if the pressure at port **B** relative to a hydraulic reference at atmospheric pressure is less than the value specified in the **Valve pressure setting** block parameter.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve pressure setting` — Pressure drop to maintain at port B relative to a hydraulic reference

`30e5 Pa` (default) | positive scalar in units of pressure

Gauge pressure that the valve is to maintain at port **B**. The valve closes in proportion to the gauge pressure if it should exceed the value specified here. If the pressure drop is less than this value, the opening area is that specified in the **Maximum passage area** block parameter.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve regulation range` — Pressure interval over which the valve is designed to operate

`1.5e5 Pa` (default) | positive scalar in units of pressure

Pressure interval over which the valve is designed to operate. This interval stretches from the largest pressure at which the valve is fully open to the lowest pressure at which the valve is fully closed. Only within this range is the valve capable of maintaining the specified pressure setting.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates

`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Leakage area` — Opening area of the valve in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Pressure differential vector` — Vector of pressure drops at which to tabulate the valve opening areas

`[30.15E5 : 15E3 : 31.35E5] Pa` (default) | vector of positive numbers in units of pressure

Vector of gauge pressures at port **B** at which to specify the opening area of the valve. The vector elements must increase monotonically from left to right. This order is important when specifying the **Opening area vector** block parameter.

The block uses this data to construct a lookup table by which to determine from the gauge pressure at port **B** the valve opening area. Data is handled with linear interpolation

(within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Opening area vector` — Vector of opening areas at the specified pressure differential breakpoints

`[9E-5 : -1 : 1E-5] m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Pressure differential vector** block parameter. The vector elements must decrease monotonically from left to right (with increasing pressure). For best results, avoid regions of flattened slope.

The block uses this data to construct a lookup table by which to determine from the gauge pressure at port **B** the valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Laminar transition specification` — Parameter in terms of which to define the boundary between laminar and turbulent regimes

`Pressure ratio` (default) | `Reynolds number`

Parameter in terms of which to specify the boundary between the laminar and turbulent flow regimes. The pressure ratio of the default parameterization is defined as the gauge pressure at the outlet divided by the same at the inlet.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow is assumed to transition between laminar and turbulent regimes. The pressure ratio is defined as the gauge pressure at the outlet divided by the same at the inlet. The transition is assumed to be smooth and centered on this value.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Reynolds number`.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes

12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Reynolds number`.

`Opening dynamics` — Choice of whether to capture the opening dynamics of the valve

`Do not include valve opening dynamics` (default) | `Include valve opening dynamics`

Choice of whether to capture the opening dynamics of the valve. Selecting `Include valve opening dynamics` causes the valve to open gradually, so as to approach its new steady-state area over a small time span. The characteristic time for such transitions is given in the **Opening time constant** block parameter.

Setting this parameter to the alternative `Do not include valve opening dynamics` is equivalent to specifying a value of `0` for the opening time constant. The opening area is in this case assumed to reach its new steady-state value instantaneously.

Include opening dynamics to more accurately capture the behavior of a real valve. For best real-time simulation performance, use with a local solver or disable valve opening dynamics altogether.

`Opening time constant` — Characteristic time scale of the valve opening transitions

`0.1 s` (default) | positive scalar in units of time

Measure of the time taken by the valve to transition from its current opening area to a new steady-state value. The block uses this parameter to calculate the rate at which a valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

`Initial area` — Opening area at the start of simulation

`1e-12 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow within the valve at the start of simulation. The block uses this parameter to calculate the initial rate at which the valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

See Also

Pressure Compensator | Pressure Reducing 3-Way Valve | Pressure Relief Valve

Introduced in R2016a

Pressure Reducing Valve (G)

Valve for restricting flow when at excessive pressure

Library: Simscape / Fluids / Gas / Valves & Orifices / Pressure Control Valves

Description

The Pressure Reducing Valve (G) block models a valve that constricts so as to maintain a preset gauge pressure at its outlet (port **B**). The normally open valve begins to close when the outlet pressure, measured against atmospheric pressure, rises above the valve pressure setting. The opening area falls with the pressure drop down to a minimum—a point at which the valve is fully closed and only leakage flow remains.

The relationship between the opening area and the pressure drop depends on the parameterization of the valve. That relationship can take the form of a linear analytical expression or of a (generally nonlinear) tabulated function.

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Control and Other Pressures

The pressure at the outlet serves as the control signal of the valve. The greater its rise over the pressure setting of the same, the smaller its opening area becomes. For use in the block calculations, its value is defined as:

$$p_{\mathrm{Ctl}} = p_{\mathrm{B}},$$

where p is instantaneous pressure. The subscript `Ctl` denotes the control value and the subscript **B** that at the port. Both are determined during simulation as absolute pressures. The pressure setting of the valve is likewise computed as:

$$P_{\mathrm{Set}} = P_{\mathrm{B,Set}} + P_{\mathrm{Atm}},$$

where P is a constant pressure parameter. The subscript `Set` denotes the valve setting, with the subscript `B,Set` being that specified in the block as a gauge pressure at the outlet (port **B**). The subscript `Atm` denotes the atmospheric value (specified in the Gas Properties (G) block of the model). For the maximum pressure of the valve:

$$P_{\mathrm{Max}} = P_{\mathrm{Set}} + \Delta P,$$

where ΔP refers specifically to the pressure regulation range of the valve, obtained as a constant from the block parameter of the same name. The subscript `Max` denotes the maximum value.

Control Pressure Overshoot

The degree to which the control pressure exceeds the pressure setting determines how much the valve will close. The pressure overshoot is expressed here as a fraction of the (width of the) pressure regulation range:

$$\hat{p} = \frac{p_{\mathrm{Ctl}} - p_{\mathrm{Set}}}{\Delta P}.$$

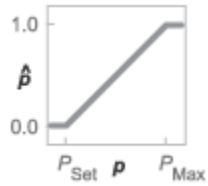
The fraction—technically, the overshoot, *normalized*—is `0` in the fully closed valve and `1` in the fully open valve. If the calculation should return a value outside of these bounds, the nearest of the two is used instead. (In other words, the fraction is *saturated* at `0` and `1`.)

The normalized (and saturated) control pressure overshoot spans three regions. Below the pressure setting of the valve, its value is a constant zero. Above the maximum pressure—the sum of the pressure setting and the pressure regulation range—it is `1`. In between, it varies, as a linear function of the control pressure measurement, p_{Ctl}.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to

recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish the simulation run, perhaps excessively so for practical use in real-time simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized pressure overshoot with sharp transitions

The block removes the discontinuities by smoothing them over a specified time scale. The smoothing, which adds a slight distortion to the control pressure overshoot, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_{\mathrm{L}} = 3\bar{p}_{\mathrm{L}}^{2} - 2\bar{p}_{\mathrm{L}}^{3}$$

and

$$\lambda_{\mathrm{R}} = 3\bar{p}_{\mathrm{R}}^{2} - 2\bar{p}_{\mathrm{R}}^{3},$$

where

$$\bar{p}_{\mathrm{L}} = \frac{\hat{p}}{\Delta p^{*}}$$

and

$$\bar{p}_{\mathrm{R}} = \frac{\hat{p} - \left(1 - \Delta p^{*}\right)}{\Delta p^{*}}.$$

In the equations:

- λ_{L} is the smoothing expression for the transition from the maximally closed position.
- λ_{R} is the smoothing expression for the transition from the fully open position.
- Δp^{*} is the (unitless) characteristic width of the pressure smoothing region:

  $$\Delta p^{*} = f^{*}\frac{1}{2},$$

  where f^{*} is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

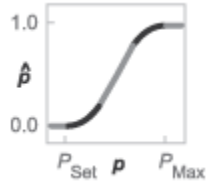
  When the smoothing factor is `0`, the control pressure overshoot stays in its original form—no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the pressure regulation range (with the control pressure overshoot taking the shape of an *S*-curve).

  At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the pressure regulation range on each side (for a total smooth region of half the regulation range).

The smoothing adds two new regions to the control pressure overshoot—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$\hat{p}^{*} = \begin{cases} 0, & \hat{p} \le 0 \\ \hat{p}\lambda_{\mathrm{L}}, & \hat{p} < \Delta P^{*} \\ \hat{p}, & \hat{p} \le 1 - \Delta P^{*} \\ \hat{p}\left(1 - \lambda_{\mathrm{R}}\right) + \lambda_{\mathrm{R}}, & \hat{p} < 1 \\ 1 & \hat{p} \ge 1 \end{cases},$$

where the asterisk indicates a smoothed variable. The figure shows the effect of smoothing on the sharpness of the transitions.



Sonic Conductance

As the normalized control pressure varies during simulation, so does the mass flow rate through the valve. The relationship between the two variables, however, is indirect. The mass flow rate is defined in terms of the valve's sonic conductance and it is this quantity that the normalized inlet pressure determines.

Sonic conductance, if you are unfamiliar with it, describes the ease with which a gas will flow when it is choked—when its velocity is at its theoretical maximum (the local speed of sound). Its measurement and calculation are covered in detail in the ISO 6358 standard (on which this block is based).

Only one value is commonly reported in valve data sheets: one taken at steady state in the fully open position. This is the same specified in the **Sonic conductance at maximum flow** parameter when the **Valve parameterization** setting is `Sonic conductance`. For values across the opening range of the valve, this maximum is scaled by the normalized pressure overshoot:

$$C = \left(C_{\mathrm{Max}} - C_{\mathrm{Min}} \right) \hat{p} + C_{\mathrm{Min}},$$

where C is sonic conductance and the subscripts `Max` and `Min` denote its values in the fully open and fully closed valve.

Because sonic conductance may not be available (or the most convenient choice for your model), the block provides several equivalent parameterizations. Use the **Valve parameterization** drop-down list to select the best for the data at hand. The parameterizations are:

- `Restriction area`
- `Sonic conductance`
- `Cv coefficient (USCS)`

- `Kv coefficient (SI)`

The parameterizations differ only in the data that they require of you. Their mass flow rate calculations are still based on sonic conductance. If you select a parameterization other than `Sonic conductance`, then the block converts the alternate data—the (computed) opening area or a (specified) flow coefficient—into an equivalent sonic conductance.

The flow coefficients measure what is, at bottom, the same quantity—the flow rate through the valve at some agreed-upon temperature and pressure differential. They differ only in the standard conditions used in their definition and in the physical units used in their expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Cv coefficient (USCS)`.
- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `1 bar`; it is expressed in metric units of m^3/h. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Kv coefficient (SI)`.

If the valve parameterization is set to `Cv Coefficient (USCS)`, the sonic conductance is computed at the maximally closed and fully open valve positions from the **Cv coefficient (SI) at maximum flow** and **Cv coefficient (SI) at leakage flow** block parameters:

$$C = \left(4 \times 10^{-8} C_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where C_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`. (These are used in the mass flow rate calculations given in the Momentum Balance section.)

If the `Kv coefficient (SI)` parameterization is used instead, the sonic conductance is computed at the same valve positions (maximally closed and fully open) from the **Kv coefficient (USCS) at maximum flow** and **Kv coefficient (USCS) at leakage flow** block parameters:

$$C = \left(4.758 \times 10^{-8} K_{\mathrm{v}}\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where K_{v} is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`.

For the `Restriction area` parameterization, the sonic conductance is computed (at the same valve positions) from the **Maximum opening area**, and **Leakage area** block parameters:

$$C = \left(0.128 \times 4S/\pi\right) \mathrm{L}/(\mathrm{s\ bar}),$$

where S is the opening area at maximum or leakage flow. The subsonic index, m, is set to `0.5` while the critical pressure ratio, b_{cr} is computed from the expression:

$$0.41 + 0.272\left[\frac{p\left(S_{\mathrm{Max}} - S_{\mathrm{Leak}}\right) + S_{\mathrm{Leak}}}{S}\right]^{0.25}.$$

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is assumed to reflect entirely in the sonic conductance of the valve (or in the data of the alternate valve parameterizations).

When the flow is choked, the mass flow rate is a function of the sonic conductance of the valve and of the thermodynamic conditions (pressure and temperature) established at the inlet. The function is linear with respect to pressure:

$$\dot{m}_{\mathrm{ch}} = C\rho_0 p_{\mathrm{in}}\sqrt{\frac{T_0}{T_{\mathrm{in}}}},$$

where:

- C is the sonic conductance inside the valve. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

- ρ is the gas density, here at standard conditions (subscript `0`), obtained from the **Reference density** block parameter.
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (`in`) or at standard conditions (`0`), the latter obtained from the **Reference temperature** block parameter.

When the flow is subsonic, and therefore no longer choked, the mass flow rate becomes a nonlinear function of pressure—both that at the inlet as well as the reduced value at the outlet. In the turbulent flow regime (with the outlet pressure contained in the back-pressure ratio of the valve), the mass flow rate expression is:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{p_{\mathrm{r}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m ,$$

where:

- p_{r} is the back-pressure ratio, or that between the outlet pressure (p_{out}) and the inlet pressure (p_{in}):

$$P_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the flow becomes choked. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

When the flow is laminar (and still subsonic), the mass flow rate expression changes to:

$$\dot{m}_{\mathrm{lam}} = C\rho_0 p_{\mathrm{in}} \left[\frac{1 - p_{\mathrm{r}}}{1 - b_{\mathrm{lam}}} \right] \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{b_{\mathrm{lam}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes (obtained from the **Laminar flow pressure ratio** block parameter). Combining the mass flow rate expressions into a single (piecewise) function, gives:

$$\dot{m} = \begin{cases} \dot{m}_{\text{lam}}, & b_{\text{lam}} \le p_{\text{r}} < 1 \\ \dot{m}_{\text{tur}}, & b_{\text{cr}} \le p_{\text{r}} < p_{\text{lam}} \\ \dot{m}_{\text{ch}}, & p_{\text{r}} < b_{\text{Cr}} \end{cases},$$

with the top row corresponding to subsonic and laminar flow, the middle row to subsonic and turbulent flow, and the bottom row to choked (and therefore sonic) flow.

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of gas can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**. Note that in this block the flow can reach but not exceed sonic speeds.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the gas and the wall that surrounds it. No work is done on or by the gas as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\text{A}} + \phi_{\text{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Valve entrance

gas

Opening through which the working fluid must enter the valve.

B — Valve entrance

gas

Opening through which the working fluid must exit the valve.

Parameters

Valve parameterization — Method by which to characterize the opening of the valve

`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the **Sonic conductance** parameterization; if a different option is selected, the data specified in converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the block description for more information on the conversion.

This parameter determines which measures of valve opening you must specify—and therefore which of those measures appear as parameters in the block dialog box.

Opening parameterization — Method by which to calculate the opening characteristics of the valve

`Linear` (default) | `Tabulated data`

Method by which to calculate the opening area of the valve. The default setting treats the opening area as a linear function of the orifice opening fraction. The alternative setting allows for a general, nonlinear relationship to be specified (in tabulated form).

Set pressure (gauge) — Inlet gauge pressure required to open the valve

`0.1 MPa` (default) | scalar in units of pressure

Inlet gauge pressure required to open the valve. This parameter marks the beginning of the pressure regulation range, over which the valve progressively opens in an effort to contain a pressure rise. The opening area of the valve depends on the difference between the actual inlet pressure to the value specified here.

Dependencies

This parameter is active and exposed in the block dialog box when the **Opening parameterization** setting is `Linear`.

`Pressure regulation range` — Width of the pressure range over which the opening area is variable

`0.1 MPa` (default) | scalar in units of pressure

Width of the pressure range over which the opening area of the valve is variable. The pressure range begins at the set pressure (specified in the block parameter of the same name). It ends at the sum of the set pressure with the value specified here. The smaller the pressure regulation range, the greater the impact of a unit pressure rise on the opening area of the valve.

Dependencies

This parameter is active and exposed in the block dialog box when the **Opening parameterization** setting is `Linear`.

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions

`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value generally reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions

`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*.

Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Maximum opening area` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position, when the valve is at the upper limit of the pressure regulation range. The block uses this parameter to scale the chosen measure of valve opening—sonic conductance, say, or C_V flow coefficient—throughout the pressure regulation range.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

Leakage area — Opening area in the maximally closed position due to sealing imperfections

`1e-12 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

`Control pressure (gauge) vector` — Vector of control pressures at which to specify valve opening data

`[0.2:0.2:1.2] MPa` (default) | vector in units of pressure

Vector of control pressures at which to specify the chosen measure of valve opening—sonic conductance, flow coefficient (in SI or USCS forms), or opening area. This vector must be equal in size to that (or those) containing the valve opening data. The vector elements must be positive and increase monotonically in value from left to right.

The first vector element gives the pressure setting of the valve (at which the valve begins to close). This element is equivalent to the **Set pressure** parameter of the `Linear` opening parameterizations. The last element gives the maximum pressure, at which the valve is maximally closed and only leakage flow remains. The difference between the two gives the pressure regulation range of the valve.

Dependencies

This parameter is active and exposed in the block dialog box when the **Opening parameterization** setting is `Tabulated data`.

`Sonic conductance vector` — Vector of sonic conductances at given control pressures

`[1e-05, .32, .64, .96, 1.28, 1.6] l/s/bar` (default) | vector with units of volume/time/pressure

Vector of sonic conductances at the given valve control pressures. The sonic conductances each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as

the ordinate. The two vectors—of sonic conductances and control pressures—must be the same in size.

Sonic conductance measures the flow rate through the fully open valve at the onset of choking (when the flow reaches its maximum velocity, the local speed of sound). Measurements are carried out at some reference inlet conditions (generally those outlined in ISO 8778). Here, those conditions include the pressures given (as interpolation breakpoints) in the control pressure vector.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Critical pressure ratio vector` — Vector of critical pressure ratios at given control pressures

`0.3 * ones(1, 6)` (default) | unitless vector

Vector of critical pressure ratios at the given valve control pressures. The ratios each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of critical pressure ratios and control pressures—must be the same in size.

The critical pressure ratio is the fraction of downstream over upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). Measurements are carried out at some reference inlet conditions (generally those outlined in ISO 8778). Here, those conditions include the pressures given (as interpolation breakpoints) in the control pressure vector.

The values specified here must each be greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Cv coefficient (USCS) vector` — Vector of flow coefficients, in USCS units, at given control pressures

`[1e-06, .08, .16, .24, .32, .4]` (default) | vector in units of ft^3/min

Vector of flow coefficients, expressed in US customary units of `ft^3/min` (as described in NFPA T3.21.3), at the given valve control pressures. The flow coefficients each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of flow coefficients and control pressures—must be the same in size.

The flow coefficient measures the relative ease with which the flow, when subjected to some reference pressure condition, can traverse the valve. Its value is commonly reported in technical data sheets for the fully open valve (alongside the pressure condition to which the measurement corresponds). The vector elements specified here each correspond to a different measurement.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)` and the **Opening parameterization** setting is `Tabulated data`.

`Kv coefficient (SI) vector` — Vector of flow coefficients, in SI units, at given control pressures

`[1e-06, .06, .12, .18, .24, .3]` (default) | vector in units of L/min

Vector of flow coefficients, expressed in SI units of `L/min` (as described in NFPA T3.21.3), at the given valve control pressures. The flow coefficients each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of flow coefficients and control pressures—must be the same in size.

The flow coefficient measures the relative ease with which the flow, when subjected to some reference pressure condition, can traverse the valve. Its value is commonly reported in technical data sheets for the fully open valve (alongside the pressure condition to which the measurement corresponds). The vector elements specified here each correspond to a different measurement.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)` and the **Opening parameterization** setting is `Tabulated data`.

`Opening area vector` — Vector of opening areas at given control pressures

`[1e-10, 2e-06, 4e-06, 6e-06, 8e-06, 1e-05] m^2` (default) | vector in units of area

Vector of valve opening areas at the given control pressure breakpoints. The opening areas each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of opening areas and control pressures—must be the same in size.

The opening area is that normal to the direction of flow at the point of narrowest aperture (the location of the valve control member).

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area` and the **Opening parameterization** setting is `Tabulated data`.

`Cross-sectional area at ports A, B, and T` — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be of the same size. The flow area specified here should (ideally) match those of the inlets of adjoining components.

`Smoothing factor` — Amount of smoothing to apply to the valve opening function

`0` (default) | positive unitless scalar

Amount of smoothing to apply to the opening function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings). These can slow down the rate of simulation.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes

`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance

`293.15` K (default) | scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance

`1.185` (default) | positive scalar in units of mass/volume

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

See Also

Pressure Relief Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

Pressure Reducing Valve (TL)

Pressure control valve for maintaining reduced pressure in fluid network portion

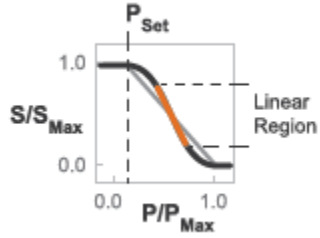
Library

Thermal Liquid/Valves & Orifices/Pressure Control Valves

Description

The Pressure Reducing Valve (TL) block represents a valve for maintaining a reduced pressure in portion of a fluid network. The valve stays fully open when the pressure at port B is lower than the valve set pressure. At the set pressure, the valve control member moves to reduce the flow rate through the valve. The valve opening area continues to decrease with increasing pressure until only leakage flow remains.

A smoothing function allows the valve opening area to change smoothly between the fully closed and fully open positions. The smoothing function does this by removing the abrupt opening area changes at the zero and maximum ball positions. The figure shows the effect of smoothing on the valve opening area curve.



Opening-Area Curve Smoothing

Mass Balance

The mass conservation equation in the valve is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- $\dot{m}_A$ is the mass flow rate into the valve through port A.
- $\dot{m}_B$ is the mass flow rate into the valve through port B.

Momentum Balance

The momentum conservation equation in the valve is

$$p_A - p_B = \frac{\dot{m}\sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{2\rho_{Avg} C_d^2 S^2}\left[1 - \left(\frac{S_R}{S}\right)^2\right] PR_{Loss},$$

where:

- p_A and p_B are the pressures at port A and port B.
- $\dot{m}$ is the mass flow rate.
- $\dot{m}_{cr}$ is the critical mass flow rate.
- ρ_{Avg} is the average liquid density.
- C_d is the discharge coefficient.
- S_R is the valve opening area.
- S is the valve inlet area.
- PR_{Loss} is the pressure ratio:

$$PR_{Loss} = \frac{\sqrt{1 - (S_R/S)^2\left(1 - C_d^2\right)} - C_d\left(S_R/S\right)}{\sqrt{1 - (S_R/S)^2\left(1 - C_d^2\right)} + C_d\left(S_R/S\right)}.$$

The valve opening area is computed as

$$S_R = \begin{cases} S_{Max}, & p_{control} \le p_{set} \\ S_{Max}\left(1-\lambda_L\right)+S_{Linear}\lambda_L, & p_{control} < p_{Min}+\Delta p_{smooth} \\ S_{Linear}, & p_{control} \le p_{Max}-\Delta p_{smooth} \\ S_{Linear}\left(1-\lambda_R\right)+S_{Leak}\lambda_R, & p_{control} < p_{Max} \\ S_{Leak}, & p_{control} \ge p_{Max} \end{cases},$$

where:

- S_{Leak} is the valve leakage area.
- S_{Linear} is the linear valve opening area:

  $$S_{Linear} = \left(\frac{S_{Leak}-S_{Max}}{p_{Max}-p_{set}}\right)\left(p_{control}-p_{set}\right)+S_{Max}$$

- S_{Max} is the maximum valve opening area.
- p_{control} is the valve control pressure:

  $$p_{control} = p_B$$

- p_{set} is the valve set pressure:

  $$p_{set} = p_{set,gauge} + p_{Atm}$$

- p_{Min} is the minimum pressure.
- p_{Max} is the maximum pressure:

  $$p_{max} = p_{set,gauge} + p_{range} + p_{Atm}$$

- Δp is the portion of the pressure range to smooth.
- λ_{L} and λ_{R} are the cubic polynomial smoothing functions

  $$\lambda_L = 3\bar{p}_L^2 - 2\bar{p}_L^3$$

  and

$$\lambda_R = 3\bar{p}_R^2 - 2\bar{p}_R^3$$

where:

$$\bar{p}_L = \frac{p_{control} - p_{set}}{\left(p_{set} + \Delta p_{smooth}\right) - p_{set}}$$

and

$$\bar{p}_R = \frac{p_B - \left(p_{max} - \Delta p_{smooth}\right)}{p_{max} - \left(p_{max} - \Delta p_{smooth}\right)}$$

The critical mass flow rate is

$$\dot{m}_{cr} = \mathrm{Re}_{cr}\,\mu_{Avg}\sqrt{\frac{\pi}{4}S_R}.$$

Energy Balance

The energy conservation equation in the valve is

$$\phi_A + \phi_B = 0,$$

where:

- ϕ_A is the energy flow rate into the valve through port A.
- ϕ_B is the energy flow rate into the valve through port B.

Parameters

Parameters Tab

Valve set pressure (gauge)

Minimum gauge pressure at port B required to actuate the valve. A pressure rise above the set pressure causes the valve to gradually close until only leakage flow remains. The default value is `0.1` MPa.

Pressure regulation range

Difference between the maximum and set pressures at port B. The valve begins to close at the set pressure. It is fully closed at the maximum pressure. The default value is `0.01` MPa.

Maximum opening area

Flow cross-sectional area in the fully open state. This state corresponds to pressures lower than the set pressure. The default value is `1e-4` MPa.

Leakage area

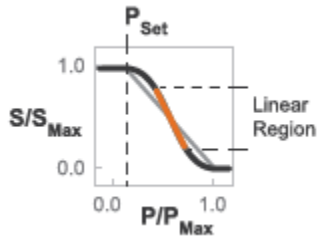
Aggregate area of all fluid leaks in the valve. The leakage area helps to prevent numerical issues due to isolated fluid network sections. For numerical robustness, set this parameter to a nonzero value. The default value is `1e-12`.

Smoothing factor

Fraction of the opening-area curve, expressed as a fraction from 0 to 1, to smooth. The block replaces the discontinuities in the opening area curve with smooth transitions that span the specified fraction of the curve. The default value is `0.01`.

A smoothing factor of 0 corresponds to a linear function that is discontinuous at the set and maximum-area pressures. A smoothing factor of 1 corresponds to a nonlinear function that changes continuously throughout the entire function domain.

A smoothing factor between 0 and 1 corresponds to a continuous piece-wise function with smooth nonlinear transitions at the set and maximum-area pressures and linear segments elsewhere.



Opening-Area Curve Smoothing

Cross-sectional area at ports A and B

Flow area at the valve inlets. The inlets are assumed equal in size. The default value is `0.01` m^2.

Characteristic longitudinal length

Approximate length of the valve. This parameter provides a measure of the longitudinal scale of the valve. The default value is `0.1` m^2.

Discharge coefficient

Semi-empirical parameter commonly used as a measure of valve performance. The discharge coefficient is defined as the ratio of the actual mass flow rate through the valve to its theoretical value.

The block uses this parameter to account for the effects of valve geometry on mass flow rates. Textbooks and valve data sheets are common sources of discharge coefficient values. By definition, all values must be greater than 0 and smaller than 1. The default value is `0.7`.

Critical Reynolds number

Reynolds number corresponding to the transition between laminar and turbulent flow regimes. The flow through the valve is assumed laminar below this value and turbulent above it. The appropriate values to use depend on the specific valve geometry. The default value is 12.

Variables Tab

Mass flow rate into port A

Mass flow rate into the component through port **A** at the start of simulation. The default value is `1 kg/s`.

Ports

- A — Thermal liquid conserving port representing valve inlet A
- B — Thermal liquid conserving port representing valve inlet B

See Also

Pressure Relief Valve (TL)

Introduced in R2016a

Pressure Relief Valve

Valve used to vent pressure in excess of a preset value from a hydraulic line

Library: Simscape / Fluids / Hydraulics (Isothermal) / Valves / Pressure Control Valves

Description

The Pressure Relief Valve block models the flow through a valve that opens to vent so as to maintain a preset pressure drop between inlet (port **A**) and outlet (port **B**). The normally closed valve cracks open when the pressure drop from **A** to **B** exceeds the valve pressure setting. The valve opening area is computed as a function of the excess in the pressure drop, either a simple linear expression or a (generally nonlinear) tabulated function. The valve serves its purpose until its opening area reaches a maximum—a point at the limit of the valve regulation range, beyond which the pressure drop is again free to rise, unabated.

Valve Opening

The opening area calculation depends on the valve parameterization selected for the block: either `Linear area-opening relationship` or `Tabulated data - Area vs. pressure`.

If the **Valve parameterization** block parameter is in the default setting of `Linear area-opening relationship`, the opening area is computed as:

$$S(\Delta p_{AB}) = S_{Leak} + k\left(\Delta p_{AB} - \Delta p_{Set}\right),$$

where:

- S_{Max} is the value specified in the **Maximum passage area** block parameter.
- Δp_{Set} is the value specified in the **Valve pressure setting** block parameter.

- Δp_{AB} is the pressure drop from port **A** to port **B**:

  $$\Delta p_{AB} = p_A - p_B,$$

  where p is the gauge pressure at the port indicated by the subscript (**A** or **B**).
- k is the linear constant of proportionality:

  $$k = \frac{S_{Max} - S_{Leak}}{\Delta p_{Reg}},$$

  where in turn:

  - S_{Leak} is the value specified in the **Leakage area** block parameter.
  - Δp_{Reg} is that specified in the **Valve regulation range** block parameter.

At and below the valve pressure setting, the opening area is that due to internal leakage alone:

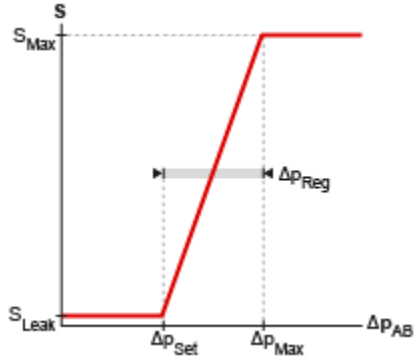
$$S(\Delta p_{AB} \le \Delta p_{Set}) = S_{Leak},$$

At and above a maximum pressure, the opening area is that of a fully open valve:

$$S(\Delta p_{AB} \ge \Delta p_{Max}) = S_{Max},$$

where the maximum pressure drop Δp_{Max} is the sum:

$$\Delta p_{Max} = \Delta p_{Set} + \Delta p_{Reg}.$$

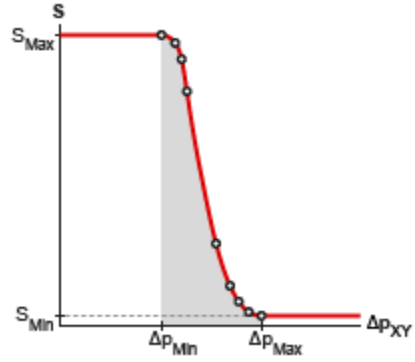


Opening area in the `Linear area-opening relationship` parameterization

If the **Valve parameterization** block parameter is set to `Tabulated data - Area vs. pressure`, the opening area is computed as:

$$S = S(\Delta p_{AB}),$$

where S_{XY} is a tabulated function constructed from the **Pressure drop vector** and **Opening area vector** block parameters. The function is based on linear interpolation (for points within the data range) and nearest-neighbor extrapolation (for points outside the data range). The leakage and maximum opening areas are the minimum and maximum values of the **Valve opening area vector** block parameter.



Opening area in the `Tabulated data - Area vs. pressure` parameterization

By default, the valve opening dynamics are ignored. The valve is assumed to respond instantaneously to changes in the pressure drop, without time lag between the onset of a pressure disturbance and the increased valve opening that the disturbance produces. If such time lags are of consequence in a model, you can capture them by setting the **Opening dynamics** block parameter to `Include valve opening dynamics`. The valves then open each at a rate given by the expression:

$$\dot{S} = \frac{S(\Delta p_{SS}) - S(\Delta p_{In})}{\tau},$$

where τ is a measure of the time needed for the instantaneous opening area (subscript `In`) to reach a new steady-state value (subscript `SS`).

The primary purpose of the leakage area of a closed valve is to ensure that at no time does a portion of the hydraulic network become isolated from the remainder of the model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage is generally present in minuscule amounts in real valves but in a model its exact value is less important than its being a small number greater than zero. The leakage area is obtained from the block parameter of the same name.

Valve Flow Rate

The causes of the pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is captured in the block by the discharge coefficient, a measure of the flow rate through the valve relative to the theoretical value that it would have in an ideal valve. The flow rate through the valve is defined as:

$$q = C_D S \sqrt{\frac{2}{\rho}} \frac{\Delta p_{AB}}{\left[(\Delta p_{AB})^2 + p_{Crit}^2 \right]^{1/4}},$$

where:

- q is the volumetric flow rate through the valve.

- C_D is the value of the **Discharge coefficient** block parameter.
- S is the opening area of the valve.
- Δp_{AB} is the pressure drop from port **A** to port **B**.
- p_{Crit} is the pressure differential at which the flow shifts between the laminar and turbulent flow regimes.

The calculation of the critical pressure depends on the setting of the **Laminar transition specification** block parameter. If this parameter is in the default setting of `By pressure ratio`:

$$p_{Crit} = \left(p_{Atm} + p_{Avg} \right) \left(1 - \beta_{Crit} \right),$$

where:

- p_{Atm} is the atmospheric pressure (as defined for the corresponding hydraulic network).
- p_{Avg} is the average of the gauge pressures at ports **A** and **B**.
- β_{Crit} is the value of the **Laminar flow pressure ratio** block parameter.

If the **Laminar transition specification** block parameter is instead set to `By Reynolds number`:

$$p_{Crit} = \frac{\rho}{2} \left(\frac{\mathrm{Re}_{Crit} \nu}{C_D D_H} \right)^2,$$

where:

- Re_{Crit} is the value of the **Critical Reynolds number** block parameter.
- ν is the kinematic viscosity specified for the hydraulic network.
- D_H is the instantaneous hydraulic diameter:

$$D_H = \sqrt{\frac{4S}{\pi}}.$$

Ports

Conserving

A — Valve inlet
hydraulic (isothermal liquid)

Opening through which the flow can enter the valve.

B — Valve outlet
hydraulic (isothermal liquid)

Opening through which the flow can exit the valve.

Parameters

Opening area parameterization — Method of calculating the opening area of the valve
`Linear area-opening relationship` (default) | `Tabulated data - Area vs. pressure`

Method by which to calculate the opening area of the valve. The default setting prescribes a linear relationship between the opening area of the valve and the pressure drop between its sensor ports. The alternative setting allows for a general, nonlinear relationship to be specified in tabulated form.

Maximum passage area — Opening area of the valve in the fully open position
`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position. The valve is fully open if the pressure drop from port **A** to port **B** is in excess of that given by the sum of the **Valve pressure setting** and **Valve regulation range** block parameters.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

Valve pressure setting — Pressure drop to maintain from port A to port B
`30e5 Pa` (default) | positive scalar in units of pressure

Pressure drop from port **A** to port **B** that the valve is to maintain. The valve begins to open when the pressure drop exceeds the value specified here. If the pressure drop is less than this value, the valve is maximally closed and the opening area is that given in the **Leakage area** block parameter.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Valve regulation range` — Pressure drop interval over which the valve is to designed to operate

`1.5e5 Pa` (default) | positive scalar in units of pressure

Pressure drop interval over which the valve is designed to operate. This interval stretches from the largest pressure drop at which the valve is maximally closed to the lowest pressure drop at which the valve is fully open. Only within this range is the valve capable of maintaining the specified pressure setting.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Flow Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates

`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the valve to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the valve: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Leakage area` — Opening area of the valve in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Linear area-pressure relationship`.

`Pressure differential vector` — Vector of pressure drops at which to tabulate the valve opening areas

`[30.15E5 : 15E3 : 31.35E5] Pa` (default) | vector of positive numbers in units of pressure

Vector of pressure drops from port **A** to port **B** at which to specify the opening area of the valve. The vector elements must increase monotonically from left to right. This order is important when specifying the **Opening area vector** block parameter.

The block uses this data to construct a lookup table by which to determine from the gauge pressure at port **B** the valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Opening area vector` — Vector of opening areas at the specified pressure differential breakpoints

`[9E-5 : -1 : 1E-5] m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Pressure differential vector** block parameter. The vector elements must increase monotonically from left to right (the direction of increasing pressure). For best results, avoid regions of flattened slope.

The block uses this data to construct a lookup table by which to determine from the pressure differential the valve opening area. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the range).

Dependencies

This parameter is active when the **Opening area parameterization** block parameter is set to `Tabulated data - Area vs. pressure`.

`Laminar transition specification` — Parameter in terms of which to define the boundary between laminar and turbulent regimes
`Pressure ratio` (default) | `Reynolds number`

Parameter in terms of which to specify the boundary between the laminar and turbulent flow regimes. The pressure ratio of the default parameterization is defined as the gauge pressure at the outlet divided by the same at the inlet.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes
`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow is assumed to transition between laminar and turbulent regimes. The pressure ratio is defined as the gauge pressure at the outlet divided by the same at the inlet. The transition is assumed to be smooth and centered on this value.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Reynolds number`.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes
12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

Dependencies

This parameter is active when the **Laminar transition specification** block parameter is set to `Reynolds number`.

`Opening dynamics` — Choice of whether to capture the opening dynamics of the valve
`Do not include valve opening dynamics` (default) | `Include valve opening dynamics`

Choice of whether to capture the opening dynamics of the valve. Selecting `Include valve opening dynamics` causes the valve to open gradually, so as to approach its new steady-state area over a small time span. The characteristic time for such transitions is given in the **Opening time constant** block parameter.

Setting this parameter to the alternative `Do not include valve opening dynamics` is equivalent to specifying a value of `0` for the opening time constant. The opening area is in this case assumed to reach its new steady-state value instantaneously.

Include opening dynamics to more accurately capture the behavior of a real valve. For best real-time simulation performance, use with a local solver or disable valve opening dynamics altogether.

`Opening time constant` — Characteristic time scale of the valve opening transitions

`0.1 s` (default) | positive scalar in units of time

Measure of the time taken by the valve to transition from its current opening area to a new steady-state value. The block uses this parameter to calculate the rate at which a valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

`Initial area` — Opening area at the start of simulation

`1e-12 m^2` (default) | positive scalar in units of area

Area normal to the direction of flow within the valve at the start of simulation. The block uses this parameter to calculate the initial rate at which the valve is opening and from it the instantaneous opening area at the next time step.

Dependencies

This parameter is active when the **Opening dynamics** block parameter is set to `Include valve opening dynamics`.

See Also

Pressure Compensator | Pressure Reducing 3-Way Valve | Pressure Reducing Valve

Introduced in R2016a

Pressure Relief Valve (G)

Valve for venting fluid when at excessive pressure

Library: Simscape / Fluids / Gas / Valves & Orifices / Pressure Control Valves

Description

The Pressure Relief Valve (G) block models an orifice that opens with inlet pressure so as to keep its level from reaching extreme levels. At normal pressures, the valve is closed and only leakage flow develops. Above a specified pressure setting, the valve begins to open, allowing the gas to vent from its region of high pressure. The opening area increases with inlet pressure up to a maximum value, at which point the valve is fully open and pressure is again free to rise unabated.

The valve responds to either of two inlet pressure measurements—one relative to the outlet or to the environment. The chosen pressure measurement is referred to as the *control pressure* of the valve and it impacts the calculations of the block. The **Control pressure specification** block parameter determines which of the measurements the block uses during simulation.

The relationship between the opening area and the pressure drop depends on the parameterization of the valve. That relationship can take the form of a linear analytical expression or of a (generally nonlinear) tabulated function.

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Control and Other Pressures

The pressure at the inlet serves as the control signal of the valve. The greater its rise over the valve's pressure setting, the greater the opening area becomes. The inlet pressure

measurement, however, can be relative to the outlet or the environment. The choice of reference depends on the setting of the **Pressure control specification** block parameter —either `Pressure differential` or `Pressure at port A`. The term *control pressure* is used here to refer to both measurements.

When the **Pressure control specification** parameter is set to `Pressure differential` (the default setting), the control pressure is computed as:

$$p_{\mathrm{Ctl}} = p_{\mathrm{A}} - p_{\mathrm{B}},$$

where p is instantaneous pressure. The subscript `Ctl` denotes the control value and the subscripts **A** and **B** the inlet and outlet, respectively. The port pressures are instantaneous values determined during simulation. The pressure setting is likewise defined as:

$$P_{\mathrm{Set}} = \left[P_{\mathrm{A}} - P_{\mathrm{B}}\right]_{\mathrm{Set}},$$

where P is a constant pressure parameter. The subscript `Set` denotes the valve setting (here a differential). The term in parentheses is obtained as a constant from the **Set pressure differential** block parameter. Similarly for the maximum pressure of the valve (at which the valve is fully open):

$$P_{\mathrm{Max}} = P_{\mathrm{Set}} + \Delta P,$$

where ΔP refers specifically to the pressure regulation range of the valve, obtained as a constant from the block parameter of the same name. The subscript `Max` denotes the maximum value.

When the **Pressure control specification** parameter is set to `Pressure at port A`, the control pressure is computed as:

$$p_{\mathrm{Ctl}} = p_{\mathrm{A}} + p_{\mathrm{Atm}},$$

where the subscript `Atm` denotes the atmospheric value (specified for the model as a constant in the Gas Properties (G) block of the Simscape Foundation library). The port pressure is an instantaneous value determined during simulation. For the pressure setting:

$$P_{\mathrm{Set}} = P_{\mathrm{A,Set}} + P_{\mathrm{Atm}},$$

where the subscript `A,Set` denotes the valve setting, specified as a gauge pressure at port **A**. This value is obtained as a constant from the **Set pressure (gauge)** block parameter. The maximum pressure of the valve is:

$$P_{\mathrm{Max}} = P_{\mathrm{Set}} + \Delta P,$$

(where, as before, ΔP refers specifically to the pressure regulation range of the valve).

Control Pressure Overshoot

The degree to which the control pressure exceeds the pressure setting determines how much the valve will open. The pressure overshoot is expressed here as a fraction of the (width of the) pressure regulation range:

$$\hat{p} = \frac{p_{\mathrm{Ctl}} - p_{\mathrm{Set}}}{\Delta P}.$$

The control pressure (p_{Ctl}), pressure setting (p_{Set}), and pressure regulation range (ΔP) are those defined for the control pressure specification chosen (`Pressure differential` or `Pressure at port A`).

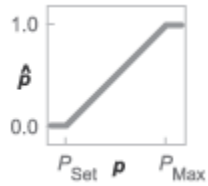
The fraction—technically, the overshoot *normalized*—is valued at `0` in the fully closed valve and `1` in the fully open valve. If the calculation should return a value outside of these bounds, the nearest of the two is used instead. (In other words, the fraction *saturates* at `0` and `1`.)

The normalized (and saturated) control pressure overshoot spans three regions. Below the pressure setting of the valve, its value is a constant zero. Above the maximum pressure—the sum of the pressure setting and the pressure regulation range—it is `1`. In between, it varies, as a linear function of the control pressure measurement, p_{Ctl}.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed

to finish the simulation run, perhaps excessively so for practical use in real-time simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized pressure overshoot with sharp transitions

The block removes the discontinuities by smoothing them over a specified time scale. The smoothing, which adds a slight distortion to the control pressure overshoot, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_{\mathrm{L}} = 3\bar{p}_{\mathrm{L}}^{2} - 2\bar{p}_{\mathrm{L}}^{3}$$

and

$$\lambda_{\mathrm{R}} = 3\bar{p}_{\mathrm{R}}^{2} - 2\bar{p}_{\mathrm{R}}^{3},$$

where

$$\bar{p}_{\mathrm{L}} = \frac{\hat{p}}{\Delta p^{*}}$$

and

$$\bar{p}_{\mathrm{R}} = \frac{\hat{p} - \left(1 - \Delta p^{*}\right)}{\Delta p^{*}}.$$

In the equations:

- λ_L is the smoothing expression for the transition from the maximally closed position.
- λ_R is the smoothing expression for the transition from the fully open position.
- Δp^* is the (unitless) characteristic width of the pressure smoothing region:

  $$\Delta p^* = f^* \frac{1}{2},$$

  where f^* is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

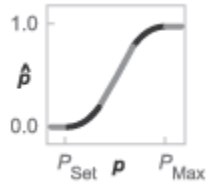
  When the smoothing factor is `0`, the control pressure overshoot stays in its original form—no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the pressure regulation range (with the control pressure overshoot taking the shape of an *S*-curve).

  At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the pressure regulation range on each side (for a total smooth region of half the regulation range).

The smoothing adds two new regions to the control pressure overshoot—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$\hat{p}^* = \begin{cases} 0, & \hat{p} \le 0 \\ \hat{p}\lambda_L, & \hat{p} < \Delta P^* \\ \hat{p}, & \hat{p} \le 1 - \Delta P^* \\ \hat{p}(1-\lambda_R) + \lambda_R, & \hat{p} < 1 \\ 1 & \hat{p} \ge 1 \end{cases},$$

where the asterisk denotes a smoothed variable. The figure shows the effect of smoothing on the sharpness of the transitions.



Sonic Conductance

As the normalized control pressure varies during simulation, so does the mass flow rate through the valve. The relationship between the two variables, however, is indirect. The mass flow rate is defined in terms of the valve's sonic conductance and it is this quantity that the normalized inlet pressure determines.

Sonic conductance, if you are unfamiliar with it, describes the ease with which a gas will flow when it is choked—when its velocity is at its theoretical maximum (the local speed of sound). Its measurement and calculation are covered in detail in the ISO 6358 standard (on which this block is based).

Only one value is commonly reported in valve data sheets: one taken at steady state in the fully open position. This is the same specified in the **Sonic conductance at maximum flow** parameter when the **Valve parameterization** setting is `Sonic conductance`. For values across the opening range of the valve, this maximum is scaled by the normalized pressure overshoot:

$$C = \left(C_{\mathrm{Max}} - C_{\mathrm{Min}} \right) \hat{p} + C_{\mathrm{Min}},$$

where C is sonic conductance and the subscripts `Max` and `Min` denote its values in the fully open and fully closed valve.

Because sonic conductance may not be available (or the most convenient choice for your model), the block provides several equivalent parameterizations. Use the **Valve parameterization** drop-down list to select the best for the data at hand. The parameterizations are:

- `Restriction area`
- `Sonic conductance`
- `Cv coefficient (USCS)`
- `Kv coefficient (SI)`

The parameterizations differ only in the data that they require of you. Their mass flow rate calculations are still based on sonic conductance. If you select a parameterization other than `Sonic conductance`, then the block converts the alternate data—the (computed) opening area or a (specified) flow coefficient—into an equivalent sonic conductance.

The flow coefficients measure what is, at bottom, the same quantity—the flow rate through the valve at some agreed-upon temperature and pressure differential. They differ only in the standard conditions used in their definition and in the physical units used in their expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Cv coefficient (USCS)`.
- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `1 bar`; it is expressed in metric units of m^3/h. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Kv coefficient (SI)`.

If the valve parameterization is set to `Cv Coefficient (USCS)`, the sonic conductance is computed at the maximally closed and fully open valve positions from the **Cv coefficient (SI) at maximum flow** and **Cv coefficient (SI) at leakage flow** block parameters:

$$C = \left(4 \times 10^{-8} C_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where C_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`. (These are used in the mass flow rate calculations given in the Momentum Balance section.)

If the `Kv coefficient (SI)` parameterization is used instead, the sonic conductance is computed at the same valve positions (maximally closed and fully open) from the **Kv coefficient (USCS) at maximum flow** and **Kv coefficient (USCS) at leakage flow** block parameters:

$$C = \left(4.758 \times 10^{-8} K_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where K_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`.

For the `Restriction area` parameterization, the sonic conductance is computed (at the same valve positions) from the **Maximum opening area**, and **Leakage area** block parameters:

$$C = \left(0.128 \times 4S / \pi\right) \texttt{L/(s bar)},$$

where S is the opening area at maximum or leakage flow. The subsonic index, m, is set to `0.5` while the critical pressure ratio, b_{cr} is computed from the expression:

$$0.41 + 0.272\left[\frac{p\left(S_{\text{Max}} - S_{\text{Leak}}\right) + S_{\text{Leak}}}{S}\right]^{0.25}.$$

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is assumed to reflect entirely in the sonic conductance of the valve (or in the data of the alternate valve parameterizations).

When the flow is choked, the mass flow rate is a function of the sonic conductance of the valve and of the thermodynamic conditions (pressure and temperature) established at the inlet. The function is linear with respect to pressure:

$$\dot{m}_{\text{ch}} = C\rho_0 p_{\text{in}} \sqrt{\frac{T_0}{T_{\text{in}}}},$$

where:

- C is the sonic conductance inside the valve. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- ρ is the gas density, here at standard conditions (subscript `0`), obtained from the **Reference density** block parameter.
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (`in`) or at standard conditions (`0`), the latter obtained from the **Reference temperature** block parameter.

When the flow is subsonic, and therefore no longer choked, the mass flow rate becomes a nonlinear function of pressure—both that at the inlet as well as the reduced value at the

outlet. In the turbulent flow regime (with the outlet pressure contained in the back-pressure ratio of the valve), the mass flow rate expression is:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}} \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{p_{\mathrm{r}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m ,$$

where:

- p_{r} is the back-pressure ratio, or that between the outlet pressure (p_{out}) and the inlet pressure (p_{in}):

  $$P_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the flow becomes choked. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

When the flow is laminar (and still subsonic), the mass flow rate expression changes to:

$$\dot{m}_{\mathrm{lam}} = C\rho_0 p_{\mathrm{in}} \left[\frac{1 - p_{\mathrm{r}}}{1 - b_{\mathrm{lam}}} \right] \sqrt{\frac{T_0}{T_{\mathrm{in}}}} \left[1 - \left(\frac{b_{\mathrm{lam}} - b_{\mathrm{cr}}}{1 - b_{\mathrm{cr}}} \right)^2 \right]^m$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes (obtained from the **Laminar flow pressure ratio** block parameter). Combining the mass flow rate expressions into a single (piecewise) function, gives:

$$\dot{m} = \begin{cases} \dot{m}_{\mathrm{lam}}, & b_{\mathrm{lam}} \le p_{\mathrm{r}} < 1 \\ \dot{m}_{\mathrm{tur}}, & b_{\mathrm{cr}} \le p_{\mathrm{r}} < p_{\mathrm{lam}} \\ \dot{m}_{\mathrm{ch}}, & p_{\mathrm{r}} < b_{\mathrm{Cr}} \end{cases},$$

with the top row corresponding to subsonic and laminar flow, the middle row to subsonic and turbulent flow, and the bottom row to choked (and therefore sonic) flow.

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of gas can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**. Note that in this block the flow can reach but not exceed sonic speeds.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the gas and the wall that surrounds it. No work is done on or by the gas as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Valve entrance

gas

Opening through which the working fluid must enter the valve.

B — Valve entrance

gas

Opening through which the working fluid must exit the valve.

Parameters

`Pressure control specification` — Choice of pressure measurement to use as valve control signal

`Pressure differential` (default) | `Pressure at port A`

Choice of pressure measurement to use as valve control signal. The block uses this setting to determine when the valve should begin to open. In the default setting (`Pressure differential`), the opening pressure of the valve is expressed as a pressure drop from inlet to outlet. In the alternative setting (`Pressure at port A`), it is expressed as a gauge inlet pressure. This parameter determines which of the two measures appears as a parameter in the block dialog box.

`Set pressure differential` —

`0 MPa` (default) | positive scalar in units of pressure

Pressure drop, from inlet to outlet, required to open the valve. This value marks the beginning of the pressure regulation range, over which the valve progressively opens in an effort to contain a pressure rise. The opening area of the valve depends on the difference between the actual pressure drop to the value specified here.

Dependencies

This parameter is active and exposed in the block dialog box when the **Pressure control specification** parameter is set to `Pressure differential`.

`Set pressure (gauge)` — Inlet gauge pressure required to open the valve

`0.1 MPa` (default) | positive scalar in units of pressure

Inlet gauge pressure required to open the valve. This parameter marks the beginning of the pressure regulation range, over which the valve progressively opens in an effort to contain a pressure rise The opening area of the valve depends on the difference between the actual inlet pressure to the value specified here.

Dependencies

This parameter is active and exposed in the block dialog box when the **Pressure control specification** parameter is set to `Pressure at port A`.

`Pressure regulation range` — Width of the pressure range over which the opening area is variable

`0.1 MPa` (default) | positive scalar in units of pressure

Width of the pressure range over which the opening area of the valve is variable. The pressure range begins at the set pressure measurement specified in the block (`Set pressure differential` or **Pressure at port A**), depending on the setting of the **Pressure control specification** parameter). It ends at the sum of the set pressure with the value specified here. The smaller the pressure regulation range, the faster the valve reaches its maximum opening area.

`Valve parameterization` — Method by which to characterize the opening of the valve

`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the **Sonic conductance** parameterization; if a different option is selected, the data specified in converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the block description for more information on the conversion.

This parameter determines which measures of valve opening you must specify—and therefore which of those measures appear as parameters in the block dialog box.

`Opening parameterization` — Method by which to calculate the opening characteristics of the valve

`Linear` (default) | `Tabulated data`

Method by which to calculate the opening area of the valve. The default setting treats the opening area as a linear function of the orifice opening fraction. The alternative setting allows for a general, nonlinear relationship to be specified (in tabulated form).

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions

`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value generally reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions

`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Maximum opening area` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position, when the valve is at the upper limit of the pressure regulation range. The block uses this parameter to scale the chosen measure

of valve opening—sonic conductance, say, or C_V flow coefficient—throughout the pressure regulation range.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

Leakage area — Opening area in the maximally closed position due to sealing imperfections

`1e-12 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area`.

Control pressure (gauge) vector — Vector of control pressures at which to specify valve opening data

`[0.2:0.2:1.2] MPa` (default) | vector in units of pressure

Vector of control pressures at which to specify the chosen measure of valve opening—sonic conductance, flow coefficient (in SI or USCS forms), or opening area. This vector must be equal in size to that (or those) containing the valve opening data. The vector elements must be positive and increase monotonically in value from left to right.

The first vector element gives the pressure setting of the valve (at which the valve begins to open). This element is equivalent to the **Set pressure** parameter of the `Linear` opening parameterizations. The last element gives the maximum pressure, at which the valve is fully open. The difference between the two gives the pressure regulation range of the valve.

The definition of control pressure depends on the setting of the **Pressure control specification** parameter. It is either the pressure drop from inlet to outlet (`Pressure differential` setting) or the gauge pressure at the inlet (`Pressure at port A` setting).

Dependencies

This parameter is active and exposed in the block dialog box when the **Opening parameterization** setting is `Tabulated data`.

`Sonic conductance vector` — Vector of sonic conductances at given control pressures

`[1e-05, .32, .64, .96, 1.28, 1.6] l/s/bar` (default) | vector with units of volume/time/pressure

Vector of sonic conductances at the given valve control pressures. The sonic conductances each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of sonic conductances and control pressures—must be the same in size.

Sonic conductance measures the flow rate through the fully open valve at the onset of choking (when the flow reaches its maximum velocity, the local speed of sound). Measurements are carried out at some reference inlet conditions (generally those outlined in ISO 8778). Here, those conditions include the pressures given (as interpolation breakpoints) in the control pressure vector.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Critical pressure ratio vector` — Vector of critical pressure ratios at given control pressures

`0.3 * ones(1, 6)` (default) | unitless vector

Vector of critical pressure ratios at the given valve control pressures. The ratios each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of critical pressure ratios and control pressures—must be the same in size.

The critical pressure ratio is the fraction of downstream over upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). Measurements are carried out at some reference inlet conditions (generally those outlined in ISO 8778). Here, those conditions include the pressures given (as interpolation breakpoints) in the control pressure vector.

The values specified here must each be greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance` and the **Opening parameterization** setting is `Tabulated data`.

`Cv coefficient (USCS) vector` — Vector of flow coefficients, in USCS units, at given control pressures

`[1e-06, .08, .16, .24, .32, .4]` (default) | vector in units of ft^3/min

Vector of flow coefficients, expressed in US customary units of ft^3/min (as described in NFPA T3.21.3), at the given valve control pressures. The flow coefficients each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of flow coefficients and control pressures—must be the same in size.

The flow coefficient measures the relative ease with which the flow, when subjected to some reference pressure condition, can traverse the valve. Its value is commonly reported in technical data sheets for the fully open valve (alongside the pressure condition to which the measurement corresponds). The vector elements specified here each correspond to a different measurement.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)` and the **Opening parameterization** setting is `Tabulated data`.

`Kv coefficient (SI) vector` — Vector of flow coefficients, in SI units, at given control pressures

`[1e-06, .06, .12, .18, .24, .3]` (default) | vector in units of L/min

Vector of flow coefficients, expressed in SI units of `L/min` (as described in NFPA T3.21.3), at the given valve control pressures. The flow coefficients each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of flow coefficients and control pressures—must be the same in size.

The flow coefficient measures the relative ease with which the flow, when subjected to some reference pressure condition, can traverse the valve. Its value is commonly reported

in technical data sheets for the fully open valve (alongside the pressure condition to which the measurement corresponds). The vector elements specified here each correspond to a different measurement.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)` and the **Opening parameterization** setting is `Tabulated data`.

`Opening area vector` — Vector of opening areas at given control pressures

`[1e-10, 2e-06, 4e-06, 6e-06, 8e-06, 1e-05] m^2` (default) | vector in units of area

Vector of valve opening areas at the given control pressure breakpoints. The opening areas each correspond to an element in the **Control pressure (gauge) vector** parameter, with the latter serving (in a Cartesian plot) as the abscissa and the former as the ordinate. The two vectors—of opening areas and control pressures—must be the same in size.

The opening area is that normal to the direction of flow at the point of narrowest aperture (the location of the valve control member).

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Restriction area` and the **Opening parameterization** setting is `Tabulated data`.

`Cross-sectional area at ports A, and B` — Area normal to the flow path at the valve ports

`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be of the same size. The flow area specified here should (ideally) match those of the inlets of adjoining components.

`Smoothing factor` — Amount of smoothing to apply to the valve opening function

`0` (default) | positive unitless scalar

Amount of smoothing to apply to the opening function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings). These can slow down the rate of simulation.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent regimes
`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance
`293.15 K` (default) | scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance
`1.185` (default) | positive scalar in units of mass/volume

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

See Also

Pressure Reducing Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

Pressure Relief Valve (TL)

Pressure control valve for maintaining preset pressure in fluid network

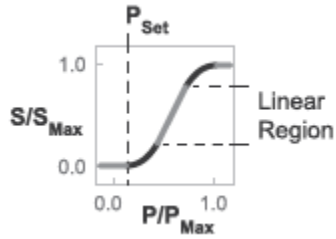
Library

Thermal Liquid/Valves & Orifices/Pressure Control Valves

Description

The Pressure Relief Valve (TL) block represents a valve for maintaining a preset pressure in a fluid network. The valve remains closed until the pressure at port A reaches the valve set pressure. A pressure rise above the set pressure causes the valve to gradually open, allowing the fluid network to relieve excess pressure.

A smoothing function allows the valve opening area to change smoothly between the fully closed and fully open positions. The smoothing function does this by removing the abrupt opening area changes at the zero and maximum ball positions. The figure shows the effect of smoothing on the valve opening area curve.



Opening-Area Curve Smoothing

Mass Balance

The mass conservation equation in the valve is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- $\dot{m}_A$ is the mass flow rate into the valve through port A.
- $\dot{m}_B$ is the mass flow rate into the valve through port B.

Momentum Balance

The momentum conservation equation in the valve is

$$p_A - p_B = \frac{\dot{m}\sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{2\rho_{Avg} C_d^2 S^2}\left[1 - \left(\frac{S_R}{S}\right)^2\right] PR_{Loss},$$

where:

- p_A and p_B are the pressures at port A and port B.
- $\dot{m}$ is the mass flow rate.
- $\dot{m}_{cr}$ is the critical mass flow rate.
- ρ_{Avg} is the average liquid density.
- C_d is the discharge coefficient.
- S_R is the valve opening area.
- S is the valve inlet area.
- PR_{Loss} is the pressure ratio:

$$PR_{Loss} = \frac{\sqrt{1 - \left(S_R/S\right)^2\left(1 - C_d^2\right)} - C_d\left(S_R/S\right)}{\sqrt{1 - \left(S_R/S\right)^2\left(1 - C_d^2\right)} + C_d\left(S_R/S\right)}.$$

The valve opening area is computed as

$$S_R = \begin{cases} S_{Leak}, & p_{control} \le p_{set} \\ S_{Leak}\left(1-\lambda_L\right) + S_{Linear}\lambda_L, & p_{control} < p_{Min} + \Delta p_{smooth} \\ S_{Linear}, & p_{control} \le p_{Max} - \Delta p_{smooth} \\ S_{Linear}\left(1-\lambda_R\right) + S_{Max}\lambda_R, & p_{control} < p_{Max} \\ S_{Max}, & p_{control} \ge p_{Max} \end{cases},$$

where:

- S_{Leak} is the valve leakage area.
- S_{Linear} is the linear valve opening area:

$$S_{Linear} = \left(\frac{S_{Max} - S_{Leak}}{p_{Max} - p_{set}}\right)\left(p_{control} - p_{set}\right) + S_{Leak}$$

- S_{Max} is the maximum valve opening area.
- p_{control} is the valve control pressure:

$$p_{control} = \begin{cases} p_A, & \text{Pressure at port A} \\ p_A - p_B, & \text{Pressure differential} \end{cases}$$

- p_{set} is the valve set pressure:

$$p_{set} = \begin{cases} p_{set,gauge} + p_{Atm}, & \text{Pressure at port A} \\ p_{set,diff}, & \text{Pressure differential} \end{cases}$$

- p_{Min} is the minimum pressure.
- p_{Max} is the maximum pressure:

$$p_{max} = \begin{cases} p_{set,gauge} + p_{range} + p_{Atm}, & \text{Pressure at port A} \\ p_{set,diff} + p_{range}, & \text{Pressure differential} \end{cases}$$

- Δp is the portion of the pressure range to smooth.
- λ_{L} and λ_{R} are the cubic polynomial smoothing functions

$$\lambda_L = 3\bar{p}_L^2 - 2\bar{p}_L^3$$

and

$$\lambda_R = 3\bar{p}_R^2 - 2\bar{p}_R^3$$

where:

$$\bar{p}_L = \frac{p_{control} - p_{set}}{\left(p_{set} + \Delta p_{smooth}\right) - p_{set}}$$

and

$$\bar{p}_R = \frac{p_{control} - \left(p_{max} - \Delta p_{smooth}\right)}{p_{max} - \left(p_{max} - \Delta p_{smooth}\right)}$$

The critical mass flow rate is

$$\dot{m}_{cr} = \mathrm{Re}_{cr}\, \mu_{Avg} \sqrt{\frac{\pi}{4} S_R}.$$

Energy Balance

The energy conservation equation in the valve is

$$\phi_A + \phi_B = 0,$$

where:

- ϕ_A is the energy flow rate into the valve through port A.
- ϕ_B is the energy flow rate into the valve through port B.

Parameters

Parameters Tab

Pressure control specification

Specification method for the valve set pressure parameter. Options include `Pressure at port A` and `Pressure differential`.

Valve set pressure (gauge)

Minimum gauge pressure at port A required to open the valve. A pressure rise above the set pressure causes the valve to gradually open until it reaches the fully open state. This parameter is active only when the **Pressure control specification** parameter is set to `Pressure at port A`. The default value is `0.1` MPa.

Valve set pressure differential

Minimum pressure differential between ports A and B required to open the valve. A pressure differential rise above this value causes the valve to gradually open until it reaches the fully open state. This parameter is active only when the **Pressure control specification** parameter is set to `Pressure differential`. The default value is `0` MPa.

Pressure regulation range

Difference between the maximum and set pressures at port A. The valve begins to open at the set pressure. It is fully open at the maximum pressure. The default value is `0.01` MPa.

Maximum opening area

Flow cross-sectional area in the fully open state. This state corresponds to pressures lower than the set pressure. The default value is `1e-4` MPa.

Leakage area

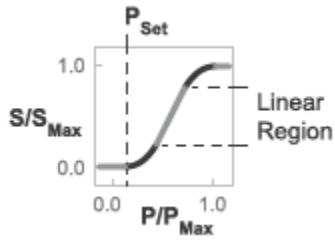
Aggregate area of all fluid leaks in the valve. The leakage area helps to prevent numerical issues due to isolated fluid network sections. For numerical robustness, set this parameter to a nonzero value. The default value is `1e-12`.

Smoothing factor

Fraction of the opening-area curve, expressed as a fraction from 0 to 1, to smooth. The block replaces the discontinuities in the opening area curve with smooth transitions that span the specified fraction of the curve. The default value is `0.01`.

A smoothing factor of 0 corresponds to a linear function that is discontinuous at the set and maximum-area pressures. A smoothing factor of 1 corresponds to a nonlinear function that changes continuously throughout the entire function domain.

A smoothing factor between 0 and 1 corresponds to a continuous piece-wise function with smooth nonlinear transitions at the set and maximum-area pressures and linear segments elsewhere.



Opening-Area Curve Smoothing

Cross-sectional area at ports A and B

Flow area at the valve inlets. The inlets are assumed equal in size. The default value is `0.01` m^2.

Characteristic longitudinal length

Approximate length of the valve. This parameter provides a measure of the longitudinal scale of the valve. The default value is `0.1` m^2.

Discharge coefficient

Semi-empirical parameter commonly used as a measure of valve performance. The discharge coefficient is defined as the ratio of the actual mass flow rate through the valve to its theoretical value.

The block uses this parameter to account for the effects of valve geometry on mass flow rates. Textbooks and valve data sheets are common sources of discharge coefficient values. By definition, all values must be greater than 0 and smaller than 1. The default value is `0.7`.

Critical Reynolds number

Reynolds number corresponding to the transition between laminar and turbulent flow regimes. The flow through the valve is assumed laminar below this value and

turbulent above it. The appropriate values to use depend on the specific valve geometry. The default value is `12`.

Variables Tab

Mass flow rate into port A

Mass flow rate into the component through port **A** at the start of simulation. The default value is `1 kg/s`.

Ports

- A — Thermal liquid conserving port representing valve inlet A
- B — Thermal liquid conserving port representing valve inlet B

See Also

Pressure Reducing Valve (TL)

Introduced in R2016a

Proportional and Servo-Valve Actuator

Continuous valve driver with output proportional to input signal

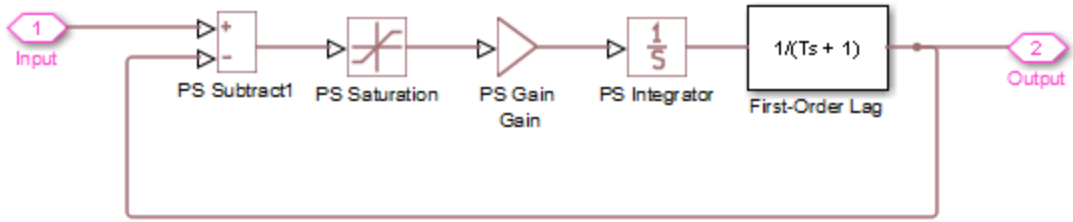
Library

Valve Actuators

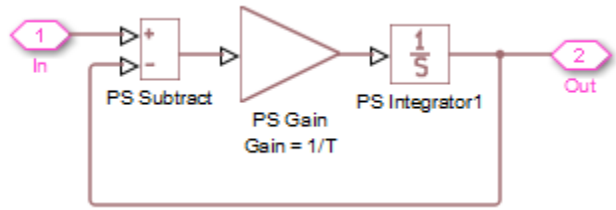
Description

The Proportional and Servo-Valve Actuator block represents an electromagnetic actuator that is used in proportional and servo-valves to drive a spool or other working member. The block is intended to work with one of the directional valve models to form a desirable configuration of a proportional or servo-valve. The block is implemented as a data-sheet-based model and reproduces only the input/output relationship, or the actuator's transient response, as presented in the catalog or data sheet.

The Proportional and Servo-Valve Actuator block is built using the blocks from the Simscape Physical Signals library. Both the input and the output of the block are physical signals. The block diagram of the model is shown in the following figure.



The model of the actuator is built of the Foundation Library Physical Signals blocks: PS Subtract, PS Saturation, PS Gain, PS Integrator. The first-order lag is arranged as a subsystem, as shown below,

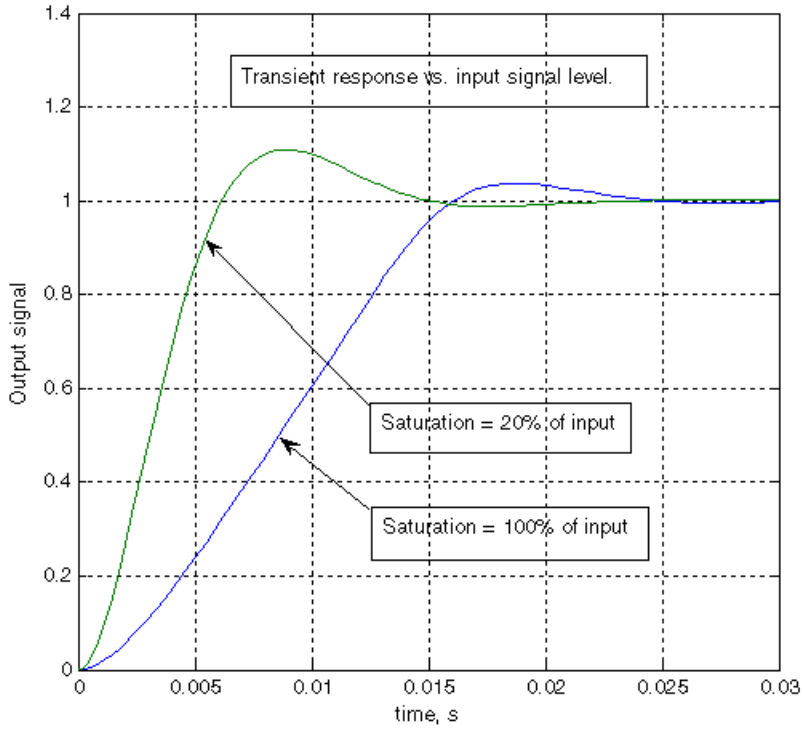


and is also built of the Physical Signals blocks: PS Subtract, PS Gain, and PS Integrator. The transfer function of the lag is

$$H(s) = \frac{1}{Ts+1}$$

where T is the **Time constant** of the lag. The value of this parameter is set through the parameter **Gain** of the PS Gain block in the First-Order Lag subsystem by setting it equal to $1/T$.

The typical transient responses of a servo-valve or a high-quality proportional valve are shown in the following figure. The only difference between the two responses in the figure is the value of the saturation.



You can adjust the block parameters, such as saturation, gain, and time constant, to make the transient responses close enough to those provided in the data sheet. The most effective way to adjust the parameters is to use the Simulink Design Optimization software.

Note This block is constructed out of blocks from the Simscape Physical Signals library (such as PS Add, PS Gain, and so on). Currently, the blocks in the Physical Signals library do not support unit propagation and checking. For more information, see “How to Work with Physical Units” (Simscape).

Parameters

Gain

Gain of the PS Gain block in the top level of the block diagram of the actuator model. The default value is `377`.

Time constant

Time constant of the first-order lag. The **Gain** parameter of the PS Gain block in the First-Order Lag subsystem is equal to 1/**Time constant**. The default value is `0.002` s.

Saturation

Saturation level of the PS Saturation block in the actuator model. The default value is `0.3`.

Ports

The block has one physical signal input port and one physical signal output port.

Examples

The Closed-Loop Actuator with Variable-Displacement Pressure-Compensated Pump example illustrates the use of the Proportional and Servo-Valve Actuator block in hydraulic systems.

See Also

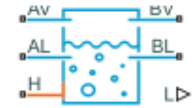
2-Position Valve Actuator | 3-Position Valve Actuator | Hydraulic Double-Acting Valve Actuator | Hydraulic Single-Acting Valve Actuator

Introduced in R2006a

Receiver Accumulator (2P)

Tank with liquid and vapor volumes of variable proportion

Library: Simscape / Fluids / Two-Phase Fluid / Tanks & Accumulators



Description

The Receiver-Accumulator (2P) block models a tank of fluid with two zones, one each for the liquid and vapor phases. The zones are distinct and the phases do not mix. They can, however, shrink and grow, one at the expense of the other, by advection and phase change. A zone can shrink to the point of disappearing—vapor wholly ceding its volume to liquid at low temperatures, for example—only to reappear when conditions reverse.

(Technically, for simulation purposes, a zone never fully disappears. A trace volume always remains inside it. This (dead) volume is roughly 1% of the total in the tank. At such minuscule levels, the zone is so small that it can be thought of as having completely vanished from the tank.)

Tanks of this sort are typical of HVAC systems—known, depending on location, as receivers or accumulators.

A typical receiver connects between a condenser and an expansion valve. The connection to the expansion valve is such that only subcooled liquid flows to it. In this block, that connection can be through the liquid ports **AL** and **BL**. As long as there is liquid in the tank, the flow out of these ports will always contain pure subcooled liquid.

An accumulator, on the other hand, normally appears between an evaporator and a compressor. The connection to the compressor is such that only superheated vapor flows to it. In this block, that connection can be through the vapor ports **AV** and **BV**. As long as there is vapor in the tank, the flow out of these ports will always contain pure superheated vapor.

The fluid can exchange heat with the tank wall—for example, to drive phase change. The thermal boundary condition at the wall is set by port **H**. The thermal mass of the wall is ignored. You can, however, capture its effects using a Thermal Mass block.

The liquid level of the tank is reported as a physical signal through port **L**. Such an indication often serves, in HVAC systems, to gauge how much refrigerant remains in the system.

Mass Balance

The tank is rigid, which is to say that its total volume is fixed during simulation. As the liquid and vapor inside can undergo phase change, however, their respective volumes, and therefore their masses also, will generally vary over time. The mass gain of one zone due to phase change is naturally the mass loss of the other. The mass balance in the liquid zone is:

$$\frac{\mathrm{d}M_{\mathrm{L}}}{\mathrm{d}t} = \dot{m}_{\mathrm{L,In}} - \dot{m}_{\mathrm{L,Out}} + \dot{m}_{\mathrm{Con}} - \dot{m}_{\mathrm{Vap}},$$

where the term on the left is the rate of mass accumulation in the liquid zone (subscript `L`) and $\dot{m}$ is the mass flow rate into or out of that zone. A minus sign indicates outflow. The subscripts `In` and `Out` denote the total mass flow rates into and out of the zone by advection. The subscripts `Con` and `Vap` denote the mass exchange rates due to condensation and vaporization at the liquid-vapor interface.

Likewise, the mass balance for the vapor zone is:

$$\frac{\mathrm{d}M_{\mathrm{V}}}{\mathrm{d}t} = \dot{m}_{\mathrm{V,In}} - \dot{m}_{\mathrm{V,Out}} - \dot{m}_{\mathrm{Con}} + \dot{m}_{\mathrm{Vap}},$$

where the subscript `V` denotes a quantity specific to the vapor zone. The mass exchange rates due to condensation and vaporization follow from energy considerations. For those calculations, see ``Liquid-Vapor Phase Change.''

The liquid phase can enter the tank through any of the ports—**AL**, **BL**, **AV**, and **BV**. So can vapor. If the fluid at a port is entirely in the liquid phase, that fluid goes to the liquid zone. If it is entirely in the vapor phase, it goes to the vapor zone. When a two-phase mixture enters the tank, then its mass fraction of liquid goes to the liquid zone and the rest to the vapor zone.

Normally—when fluid other than the dead volume remains in the zones—the liquid phase can exit the tank through ports **AL** and **BL** only; the vapor phase can flow out through

ports **AV** and **BV** only. The mass flow rate out of the liquid zone (by advection through the ports) is:

$$\dot{m}_{\mathrm{L,Out}} = -\left(\dot{m}_{\mathrm{AL}} + \dot{m}_{\mathrm{BL}}\right),$$

where the subscripts AL and BL denote what are normally the liquid ports. The mass flow rates through the individual ports are defined as positive into the tank and as negative out of the tank. The minus sign in the expression reflects the fact that the total mass flow rate out of the liquid zone is defined as positive for outflow. For the vapor zone:

$$\dot{m}_{\mathrm{V,Out}} = -\left(\dot{m}_{\mathrm{AV}} + \dot{m}_{\mathrm{BV}}\right),$$

where the subscripts AV and BV denote what are normally the vapor ports. The sign conventions are as described for the liquid zone.

If a zone shrinks to the size of its dead volume, the phase in that zone effectively disappears. The remaining phase is then considered to occupy the whole of the tank, and it is free to exit through any of the ports—including those normally used by the eliminated phase. If only liquid remains in the tank, the total mass flow rate out of the liquid zone becomes:

$$\dot{m}_{\mathrm{V,Out}} = -\left(\dot{m}_{\mathrm{AL}} + \dot{m}_{\mathrm{BL}} + \dot{m}_{\mathrm{AV}} + \dot{m}_{\mathrm{BV}}\right).$$

Conversely, if only vapor remains in the tank, the total mass flow rate out of the vapor zone becomes:

$$\dot{m}_{\mathrm{V,Out}} = -\left(\dot{m}_{\mathrm{AL}} + \dot{m}_{\mathrm{BL}} + \dot{m}_{\mathrm{AV}} + \dot{m}_{\mathrm{BV}}\right).$$

Energy Balance

The tank is not generally insulated. The liquid and vapor inside can exchange heat with the wall and, through it, with the environment. The heat exchange rate depends on the thermal boundary condition specified at port **H**. (That rate is assumed to be the same for liquid and vapor zones alike).

Heat exchange, phase change, and energy flow through the ports together account for the accumulation of specific internal energy in the tank. In the liquid zone, that accumulation (from the energy balance for that zone) is:

$$M_{\mathrm{L}}\frac{\mathrm{d}u_{\mathrm{L}}}{\mathrm{d}t}+\frac{\mathrm{d}M_{\mathrm{L}}}{\mathrm{d}t}u_{\mathrm{L}}=\phi_{\mathrm{L,In}}-\phi_{\mathrm{L,Out}}+\phi_{\mathrm{Con}}-\phi_{\mathrm{Vap}}+Q_{\mathrm{L}}.$$

where the first term on the left is the rate of accumulation of specific internal energy in the liquid zone. The second term is the rate of mass accumulation. On the right, ϕ denotes the energy flow rate into or out of the zone. As before, a minus sign indicates outflow. The subscripts are as described in ``Mass Balance.'' Q is the heat exchange rate between the tank wall and the fluid in the zone.

Likewise for the vapor volume:

$$M_{\mathrm{V}}\frac{\mathrm{d}u_{\mathrm{V}}}{\mathrm{d}t}+\frac{\mathrm{d}M_{\mathrm{V}}}{\mathrm{d}t}u_{\mathrm{V}}=\phi_{\mathrm{V,In}}-\phi_{\mathrm{V,Out}}-\phi_{\mathrm{Con}}+\phi_{\mathrm{Vap}}+Q_{\mathrm{V}}.$$

As the phases can each enter the tank through any of its ports, so can the energy that they carry. That carried by the liquid phase goes to the liquid zone. That carried by the vapor phase goes into the vapor zone. If a two-phase mixture enters the tank, its energy contents are split between the zones: the fraction carried by the liquid phase goes into the liquid zone and the remainder into the vapor zone.

When more than the dead volume remains in a zone, energy can flow out of that zone through its corresponding ports only. Energy from the liquid zone can flow out through ports **AL** and **BL** only; that from the vapor zone can flow out through ports **AV** and **BV** only. The energy flow rate out of the liquid zone by advection then is:

$$\phi_{\mathrm{L,Out}}=-\left(\phi_{\mathrm{AL}}+\phi_{\mathrm{BL}}\right),$$

As with mass flow, the energy flow rates through the individual ports are defined as positive for inflow and as negative for outflow. The minus sign in the expression reflects the fact that the total energy flow rate out of the liquid zone is defined as positive for outflow. For the vapor zone:

$$\phi_{\mathrm{V,Out}}=-\left(\phi_{\mathrm{AV}}+\phi_{\mathrm{BV}}\right),$$

If a zone contains only its dead volume, then the remaining phase is considered to occupy the tank in its entirety. The energy of that phase is then free to exit through any of the ports, even those normally used by the eliminated phase. With only liquid in the tank:

$$\phi_{\mathrm{V,Out}} = -\left(\phi_{\mathrm{AV}} + \phi_{\mathrm{BV}}\right),$$

With only vapor in the tank:

$$\phi_{\mathrm{V,Out}} = -\left(\phi_{\mathrm{AL}} + \phi_{\mathrm{BL}} + \phi_{\mathrm{AV}} + \phi_{\mathrm{BV}}\right).$$

Whether phase change occurs, and which phase transforms to which, depends on the specific enthalpy of each phase. In the liquid zone, if the average specific enthalpy is above the saturation value for the liquid phase, then liquid vaporizes. The zone cedes mass and energy to the vapor zone at a rate dependent on a phase-change time constant. If the specific enthalpy is below the saturation value, the zone loses nothing. Expressed as a piecewise function:

$$\phi_{\mathrm{Vap}} = \begin{cases} \dfrac{M_{\mathrm{L}}\left(h_{\mathrm{L}} - h_{\mathrm{L,Sat}}\right)}{\tau}, & h_{\mathrm{L}} \geq h_{\mathrm{L,Sat}} \\ 0, & h_{\mathrm{L}} < h_{\mathrm{L,Sat}} \end{cases},$$

where M_{L} is the total mass in the liquid zone and τ is the **Vaporization and condensation time constant** block parameter. The subscript `L,Sat` denotes the saturation value for the liquid phase. The energy exchange rate due to vaporization determines the mass exchange rate between the fluid zones as liquid transforms into vapor:

$$\dot{m}_{\mathrm{Vap}} = \frac{\phi_{\mathrm{Vap}}}{h_{\mathrm{V,Sat}}},$$

where the subscript `V,Sat` denotes the saturation value for the vapor phase. Implicit in this expression is the assumption that, once transformed into vapor, the liquid mass has a specific enthalpy equal to the saturation value of its new phase.

In the vapor zone, if the average specific enthalpy is below the saturation value of the vapor phase, then vapor condenses. The zone cedes mass and energy to the liquid zone at a rate dependent on the phase-change time constant specified in the block (where it is assumed to be the same for vaporization and condensation). If the specific enthalpy is above the saturation value, the zone loses nothing. Expressed as a piecewise function:

$$\phi_{\mathrm{Con}} = \begin{cases} \dfrac{M_{\mathrm{V}}\left(h_{\mathrm{V}} - h_{\mathrm{V,Sat}}\right)}{\tau}, & h_{\mathrm{V}} \geq h_{\mathrm{V,Sat}} \\ 0, & h_{\mathrm{V}} < h_{\mathrm{V,Sat}} \end{cases},$$

where M_{V} is the total mass in the vapor zone. The energy exchange rate due to condensation determines the mass exchange rate between the fluid zones as vapor transforms into liquid:

$$\dot{m}_{\mathrm{Con}} = \frac{\phi_{\mathrm{Con}}}{h_{\mathrm{L,Sat}}},$$

The vapor mass, once transformed into liquid, is assumed to be at a specific enthalpy equal to the saturation value of its new phase.

The heat exchange between the fluid and the wall is assumed to be of a convective nature. Its total rate is calculated here as the sum of the individual rates between the liquid zone and its section of the wall and, likewise, between the vapor zone and its section of the wall:

$$Q_{\mathrm{H}} = Q_{\mathrm{L}} + Q_{\mathrm{V}},$$

where Q_{H} is the total heat exchange rate arising from the thermal boundary condition specified at port **H**. Between the liquid zone and its fraction of the wall surface area:

$$Q_{\mathrm{L}} = f_{\mathrm{V,L}} S_{\mathrm{W}} \alpha_{\mathrm{L}} \left(T_{\mathrm{H}} - T_{\mathrm{L}}\right),$$

where:

- $f_{\mathrm{V,L}}$ is the volume fraction of the tank taken up by the liquid zone. The product of this fraction with the surface area of the wall gives the surface area available for heat exchange between the liquid and the tank wall.
- S_{W} is the surface available for heat exchange—that between the wall of the tank and the whole of the fluid inside.
- α_{L} is the heat transfer coefficient for the liquid zone (a parameter in the **Heat Transfer** tab of the block dialog box).

- T is the average temperature at the location denoted in the subscript—`H` for the tank wall, `L` for the liquid zone.

Between the vapor zone and its fraction of the wall surface area:

$$Q_{\mathrm{L}} = \left(1 - f_{\mathrm{V,L}}\right) S_{\mathrm{W}} \alpha_{\mathrm{V}} \left(T_{\mathrm{H}} - T_{\mathrm{V}}\right),$$

where the variables are as described for the liquid zone, but with the subscript `V` denoting a value in the vapor zone. The first term in parentheses is the volume fraction taken up by the vapor zone. The phases are each compressible and therefore of variable density. As a result, the volume fraction of a zone will generally differ from its mass fraction. The correspondence between the volume and mass fractions of the liquid zone (the only considered explicitly in the block calculations) is:

$$f_{\mathrm{V,L}} = \frac{f_{\mathrm{M,L}} \nu_{\mathrm{L}}}{f_{\mathrm{M,L}} \nu_{\mathrm{L}} + \left(1 - f_{\mathrm{M,L}}\right) \nu_{\mathrm{V}}},$$

where f`M,L` is the mass fraction of the liquid zone and ν is the specific volume of the phase indicated in the subscript (`L` for liquid, `V` for vapor).

Momentum Balance

The flow resistance due to friction and sudden changes in direction is ignored. The ports are assumed to lie on a plane and therefore at the same elevation. Pressure is therefore constant inside the tank—across both liquid and vapor zones—and at each of the ports:

$$p_{\mathrm{I}} = p_{\mathrm{AV}} = p_{\mathrm{BV}} = p_{\mathrm{AL}} = p_{\mathrm{BL}}.$$

where P is the pressure at the location indicated by the subscript. `I` denotes the interior of the tank.

Ports

Output

L — Liquid level

physical signal

Liquid level in the tank. Use this port to monitor the amount of liquid remaining inside.

Conserving

AV — Tank opening

two-phase fluid

Opening for the fluid to flow into or out of the tank. Both liquid and vapor can enter through this port. However, only vapor can exit through it—until the tank is depleted of vapor, in which event liquid too can flow out through this port.

BV — Pipe opening

two-phase fluid

Opening through which the thermal liquid flows into or out of the pipe. Ports **A** and **B** can each function as either inlet or outlet. Thermal conduction is allowed between the thermal liquid ports and the fluid internal to the pipe (though its impact is typically relevant only at near zero flow rates).

H — Tank wall

thermal

Thermal boundary between the fluid volume and the tank wall. Use this port to capture heat exchanges of various kinds—for example, conductive, convective, or radiative—between the fluid and the environment external to the tank.

Parameters

Main

Total volume — Aggregate volume of liquid and vapor phases in the tank

1 m^3 (default) | positive scalar with units of volume

Aggregate volume of liquid and vapor phases in the tank.

`Cross-sectional area at port AV` — Area normal to the direction of flow at port AV
`0.01 m^2` (default) | positive scalar with units of area

Area normal to the direction of flow at port **AV**.

`Cross-sectional area at port BV` — Area normal to the direction of flow at port BV
`0.01 m^2` (default) | positive scalar with units of area

Area normal to the direction of flow at port **BV**.

`Cross-sectional area at port AL` — Area normal to the direction of flow at port AL
`0.01 m^2` (default) | positive scalar with units of area

Area normal to the direction of flow at port **AL**.

`Cross-sectional area at port BL` — Area normal to the direction of flow at port BL
`0.01 m^2` (default) | positive scalar with units of area

Area normal to the direction of flow at port **BL**.

`Liquid volume fraction out of range` — Means by which to handle extreme volume fractions
`No error` (default) | `Error` | `Warning`

Means by which to handle unusual volume fractions. Select `Warning` to be notified when the volume fraction crosses a specified range. Select `Error` to stop simulation at such events.

`Minimum liquid volume fraction` — Value below which to trigger warning or error
`0.05` (default) | positive scalar between 0 and 1

Lower bound of the valid range for the liquid volume fraction in the tank. Fractions below this value will trigger a simulation warning or error (depending on the setting of the **Liquid volume fraction out of range** block parameter.

Dependencies

This parameter is active when the **Liquid volume fraction out of range** block parameter is set to `Warning` or `Error`.

`Maximum liquid volume fraction` — Value above which to trigger warning or error

`0.95` (default) | positive scalar between 0 and 1

Upper bound of the valid range for the liquid volume fraction in the tank. Fractions above this value will trigger a simulation warning or error (depending on the setting of the **Liquid volume fraction out of range** block parameter.

Dependencies

This parameter is active when the **Liquid volume fraction out of range** block parameter is set to `Warning` or `Error`.

`Volume fraction threshold for transition to pure liquid or pure vapor` — Volume fraction of a phase below which to transition to a single-phase tank

`20 W/(m^2*K)` (default) | positive scalar with units of power/(area*temperature)

Volume fraction of either phase below which to transition to a single-phase tank—either subcooled liquid or superheated vapor. This parameter determines how smooth the transition is. The larger its value, the smoother the transition and therefore the faster the simulation (though at the cost of lower accuracy).

Heat Transfer

`Vapor heat transfer coefficient` — Coefficient for heat exchange between vapor and the tank wall

`20 W/(m^2*K)` (default) | positive scalar with units of power/(area*temperature)

Coefficient for heat exchange between the vapor zone and its section of the tank wall. This parameter serves to calculate the rate of this heat exchange.

`Liquid heat transfer coefficient` — Coefficient for heat exchange between liquid and the tank wall

`100 W/(m^2*K)` (default) | positive scalar with units of power/(area*temperature)

Coefficient for heat exchange between the liquid zone and its section of the tank wall. This parameter serves to calculate the rate of this heat exchange.

`Total heat transfer surface area` — Wall surface area in contact with fluid
`6 m^2` (default) | positive scalar with units of area

Surface area of the tank through which heat exchange with the fluid occurs.

Effects and Initial Conditions Tab

`Initial fluid energy specification` — Thermodynamic variable whose initial value to set
`Liquid volume fraction` (default) | `Temperature` | `Liquid mass fraction` | `Specific enthalpy` | `Specific internal energy`

Thermodynamic variable in terms of which to define the initial conditions of the component.

`Initial pressure` — Absolute pressure at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure in the tank at the start of simulation, specified against absolute zero.

`Initial temperature` — Absolute temperature at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature in the tank at the start of simulation, specified against absolute zero.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Temperature`.

`Initial liquid mass fraction` — Mass fraction of liquid at the start of simulation
`0.5` (default) | unitless scalar between 0 and 1

Mass fraction of liquid in the tank at the start of simulation.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Liquid mass fraction`.

`Initial liquid volume fraction` — Volume fraction of liquid at the start of simulation
`0.5` (default) | unitless scalar between 0 and 1

Volume fraction of liquid in the tank at the start of simulation.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Liquid volume fraction`.

`Initial specific enthalpy` — Specific enthalpy at the start of simulation
`1500 kJ/kg` (default) | scalar with units of energy/mass

Specific enthalpy of the fluid in the tank at the start of simulation.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Specific enthalpy`.

`Initial specific internal energy` — Specific internal energy at the start of simulation
`1500 kJ/kg` (default) | scalar with units of energy/mass

Specific internal energy of the fluid in the tank at the start of simulation.

Dependencies

This parameter is active when the **Initial fluid energy specification** option is set to `Specific internal energy`.

`Vaporization and condensation time constant` — Characteristic duration of a phase-change event
`0.1 s` (default) | scalar with units of time

Characteristic time to equilibrium of a phase-change event taking place in the tank. Increase this parameter to slow the rate of phase change or decrease it to speed up the rate.

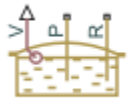
See Also

2-Port Constant Volume Chamber (2P) | 3-Port Constant Volume Chamber (2P) | 3-Zone Pipe (2P) | Constant Volume Chamber (2P)

Introduced in R2018b

Reservoir

Pressurized hydraulic reservoir



Library

Hydraulic Utilities

Description

The Reservoir block represents a pressurized hydraulic reservoir, in which fluid is stored under a specified pressure. The pressure remains constant regardless of volume change. The block accounts for pressure loss in the return line that can be caused by a filter, fittings, or some other local resistance. The loss is specified with the pressure loss coefficient. The block computes the volume of fluid in the tank and exports it outside through the physical signal port V.

The fluid volume value does not affect the results of simulation. It is introduced merely for information purposes. It is possible for the fluid volume to become negative during simulation, which signals that the fluid volume is not enough for the proper operation of the system. By viewing the results of the simulation, you can determine the extent of the fluid shortage.

Parameters

Pressurization level

The pressure inside the reservoir. The default value is 0.

Initial fluid volume

The initial volume of fluid in the tank. The default value is `0.02` m^3.

Return line diameter

The diameter of the return line. The default value is `0.02` m.

Pressure loss coefficient in return line

The value of the pressure loss coefficient, to account for pressure loss in the return line. This parameter must be greater than zero. The default value is `1`.

For reasons of computational robustness, the loss is computed with the equation similar to that used in the Fixed Orifice block:

$$q = \frac{1}{\sqrt{K}} A_p \sqrt{\frac{2}{\rho} p_{loss}}$$

The Critical Reynolds number is set to 15.

Ports

The block has the following ports:

`P`

Hydraulic conserving port associated with the pump line.

`R`

Hydraulic conserving port associated with the return line.

`V`

Physical signal port that outputs the volume of fluid in the tank.

See Also

Constant Head Tank | Hydraulic Reference | Tank

Introduced in R2006b

Resistive Pipe LP

Hydraulic pipeline which accounts for friction losses and port elevations

Library

Low-Pressure Blocks

Description

The Resistive Pipe LP block models hydraulic pipelines with circular and noncircular cross sections and accounts for resistive property only. In other words, the block is developed with the basic assumption of the steady state fluid momentum conditions. Neither fluid compressibility nor fluid inertia is considered in the model, meaning that features such as water hammer cannot be investigated. If necessary, you can add fluid compressibility, fluid inertia, and other effects to your model using other blocks, thus producing a more comprehensive model.

The end effects are also not considered, assuming that the flow is fully developed along the entire pipe length. To account for local resistances, such as bends, fittings, inlet and outlet losses, and so on, convert the resistances into their equivalent lengths, and then sum up all the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length.

Pressure loss due to friction is computed with the Darcy equation, in which losses are proportional to the flow regime-dependable friction factor and the square of the flow rate. The friction factor in turbulent regime is determined with the Haaland approximation (see [1] on page 1-1074). The friction factor during transition from laminar to turbulent regimes is determined with the linear interpolation between extreme points of the regimes. As a result of these assumptions, the tube is simulated according to the following equations:

$$p = f\frac{\left(L + L_{eq}\right)}{D_H}\frac{\rho}{2A^2}q \bullet |q| + \rho \bullet g\left(z_B - z_A\right)$$

$$f = \begin{cases} K_s / Re & \text{for } Re <= Re_L \\ f_L + \dfrac{f_T - f_L}{Re_T - Re_L}\left(Re - Re_L\right) & \text{for } Re_L < Re < Re_T \\ \dfrac{1}{\left(-1.8\log_{10}\left(\dfrac{6.9}{Re} + \left(\dfrac{r / D_H}{3.7}\right)^{1.11}\right)\right)^2} & \text{for } Re >= Re_T \end{cases}$$

$$\mathrm{Re} = \frac{q \cdot D_H}{A \cdot \nu}$$

where

p	Pressure loss along the pipe due to friction
q	Flow rate through the pipe
Re	Reynolds number
Re_L	Maximum Reynolds number at laminar flow
Re_T	Minimum Reynolds number at turbulent flow
K_s	Shape factor that characterizes the pipe cross section
f_L	Friction factor at laminar border
f_T	Friction factor at turbulent border
A	Pipe cross-sectional area
D_H	Pipe hydraulic diameter
L	Pipe geometrical length
L_{eq}	Aggregate equivalent length of local resistances

r	Height of the roughness on the pipe internal surface
ν	Fluid kinematic viscosity
z_A, z_B	Elevations of the pipe port A and port B, respectively
g	Gravity acceleration

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia, fluid compressibility, and wall compliance are not taken into account.

Parameters

- “Basic Parameters Tab” on page 1-1071
- “Vertical Position Tab” on page 1-1072
- “Variables Tab” on page 1-1073

Basic Parameters Tab

Pipe cross section type

The type of pipe cross section: `Circular` or `Noncircular`. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is `Circular`.

Internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to `Circular`. The default value is `0.01` m.

Noncircular pipe cross-sectional area

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `1e-4` m^2.

Noncircular pipe hydraulic diameter

Hydraulic diameter of the pipe cross section. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `0.0112` m.

Geometrical shape factor

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1 on page 1-1074]. The default value is `64`, which corresponds to a pipe with a circular cross section.

Pipe length

Pipe geometrical length. The default value is `5` m.

Aggregate equivalent length of local resistances

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is `1` m.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Vertical Position Tab

Port A elevation wrt reference plane

The parameter specifies vertical position of the pipe port A with respect to the reference plane. The default value is `0`.

Port B elevation wrt reference plane

The parameter specifies vertical position of the pipe port B with respect to the reference plane. The default value is `0`.

Gravitational acceleration

Value of the gravitational acceleration constant (*g*). The block uses this parameter to compute the effects of an elevation gradient between the ports on their pressure differential. The default value is `9.80655` m/s^2.

Variables Tab

Pressure drop from port A to port B

Value at time zero of the pressure drop between the ports. Simscape software uses this parameter to guide the initial configuration of this component and of the model. Initial variables that conflict with each other or that are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, lower-priority, variables.

Flow rate from port A to port B

Value at time zero of the flow rate through the ports. Simscape software uses this parameter to guide the initial configuration of this component and of the model. Initial variables that conflict with each other or that are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, lower-priority, variables.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Pipe cross section type**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Pipe cross section type** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

See Also

Hydraulic Pipe LP with Variable Elevation | Hydraulic Resistive Tube | Hydraulic Pipe LP | Hydraulic Pipeline | Linear Hydraulic Resistance | Resistive Pipe LP with Variable Elevation | Segmented Pipe LP | Segmented Pipeline

Introduced in R2009a

Resistive Pipe LP with Variable Elevation

Hydraulic pipeline which accounts for friction losses and variable port elevations

Library

Low-Pressure Blocks

Description

The Resistive Pipe LP with Variable Elevation block models hydraulic pipelines with circular and noncircular cross sections and accounts for resistive property only. Use this block for low-pressure system simulation in which the pipe ends change their positions with respect to the reference plane. The elevations are provided through respective physical signal inputs.

To reduce model complexity, you can use this block to simulate not only a pipe itself, but also a combination of pipes and local resistances such as bends, fittings, inlet and outlet losses, associated with the pipe. You must convert the resistances into their equivalent lengths, and then sum up all the resistances to obtain their aggregate length. Then add this length to the pipe geometrical length.

Pressure loss due to friction is computed with the Darcy equation, in which losses are proportional to the flow regime-dependable friction factor and the square of the flow rate. The friction factor in turbulent regime is determined with the Haaland approximation (see [1] on page 1-1080). The friction factor during transition from laminar to turbulent regimes is determined with the linear interpolation between extreme points of the regimes. As a result of these assumptions, the tube is simulated according to the following equations:

$$p = f\frac{\left(L + L_{eq}\right)}{D_H}\frac{\rho}{2A^2}q \bullet |q| + \rho \bullet g\left(z_B - z_A\right)$$

$$f = \begin{cases} K_s / Re & \text{for } Re <= Re_L \\ f_L + \dfrac{f_T - f_L}{Re_T - Re_L}\left(Re - Re_L\right) & \text{for } Re_L < Re < Re_T \\ \dfrac{1}{\left(-1.8\log_{10}\left(\dfrac{6.9}{Re} + \left(\dfrac{r / D_H}{3.7}\right)^{1.11}\right)\right)^2} & \text{for } Re >= Re_T \end{cases}$$

$$\mathrm{Re} = \frac{q \cdot D_H}{A \cdot \nu}$$

where

p	Pressure loss along the pipe due to friction
q	Flow rate through the pipe
Re	Reynolds number
Re_L	Maximum Reynolds number at laminar flow
Re_T	Minimum Reynolds number at turbulent flow
K_s	Shape factor that characterizes the pipe cross section
f_L	Friction factor at laminar border
f_T	Friction factor at turbulent border
A	Pipe cross-sectional area
D_H	Pipe hydraulic diameter
L	Pipe geometrical length
L_{eq}	Aggregate equivalent length of local resistances

r	Height of the roughness on the pipe internal surface
ν	Fluid kinematic viscosity
z_A, z_B	Elevations of the pipe port A and port B, respectively
g	Gravity acceleration

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

- Flow is assumed to be fully developed along the pipe length.
- Fluid inertia, fluid compressibility, and wall compliance are not taken into account.

Parameters

- “Basic Parameters Tab” on page 1-1077
- “Variables Tab” on page 1-1078

Basic Parameters Tab

Pipe cross section type

The type of pipe cross section: `Circular` or `Noncircular`. For a circular pipe, you specify its internal diameter. For a noncircular pipe, you specify its hydraulic diameter and pipe cross-sectional area. The default value of the parameter is `Circular`.

Internal diameter

Pipe internal diameter. The parameter is used if **Pipe cross section type** is set to `Circular`. The default value is `0.01` m.

Noncircular pipe cross-sectional area

Pipe cross-sectional area. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `1e-4` m^2.

Noncircular pipe hydraulic diameter

Hydraulic diameter of the pipe cross section. The parameter is used if **Pipe cross section type** is set to `Noncircular`. The default value is `0.0112` m.

Geometrical shape factor

Used for computing friction factor at laminar flow. The shape of the pipe cross section determines the value. For a pipe with a noncircular cross section, set the factor to an appropriate value, for example, 56 for a square, 96 for concentric annulus, 62 for rectangle (2:1), and so on [1 on page 1-1080]. The default value is `64`, which corresponds to a pipe with a circular cross section.

Pipe length

Pipe geometrical length. The default value is `5` m.

Aggregate equivalent length of local resistances

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. The default value is `1` m.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Gravitational acceleration

Value of the gravitational acceleration constant (g). The block uses this parameter to calculate time changes in pressure due to time changes in elevation. The default value is `9.80655` m/s^2.

Variables Tab

Pressure drop from port A to port B

Value at time zero of the pressure drop between the ports. Simscape software uses this parameter to guide the initial configuration of this component and of the model.

Initial variables that conflict with each other or that are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, lower-priority, variables.

Flow rate from port A to port B

Value at time zero of the flow rate through the ports. Simscape software uses this parameter to guide the initial configuration of this component and of the model. Initial variables that conflict with each other or that are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, lower-priority, variables.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the pipe inlet.

`B`

Hydraulic conserving port associated with the pipe outlet.

`el_A`

Physical signal input port that controls pipe elevation at port A.

`el_B`

Physical signal input port that controls pipe elevation at port B.

References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

See Also

Hydraulic Pipe LP with Variable Elevation | Hydraulic Pipe LP | Hydraulic Pipeline | Hydraulic Resistive Tube | Linear Hydraulic Resistance | Resistive Pipe LP | Segmented Pipe LP | Segmented Pipeline

Introduced in R2010a

Rotating Pipe

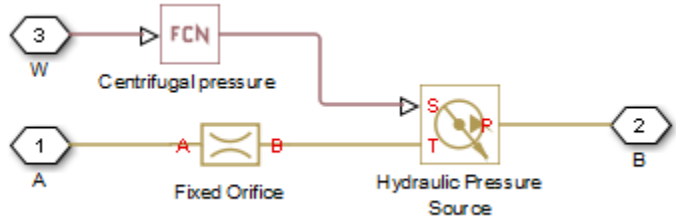
Hydraulic pipeline created by bore in rotating housing

Library

Pipelines

Description

The Rotating Pipe block models a hydraulic pipeline represented as a short bore drilled through a shaft at some angle to its axis, or as a short pipe that connects actuator or bearings, mounted on the shaft, with the channel inside the shaft. Such connections are typical for lubrication systems and for hydraulically-operated clutches and mechanisms. The shaft angular velocity is assumed to be constant or changing at very low speed. The block is implemented as a structural model, shown in the diagram below:



The Fixed Orifice block simulates pressure losses in the bore. The pipe is assumed to be short enough to be characterized by a flow-pressure relationship similar to that of the fixed orifice. The model accounts for the laminar and turbulent flow regimes by

monitoring the Reynolds number (Re) and comparing its value with the critical Reynolds number (Re_{cr}). The Hydraulic Pressure Source block represents pumping action developed by a rotating pipe, which is calculated with the equation:

$$p = \rho \cdot \frac{\omega^2 \cdot r_B^2}{2}$$

where

p	Pressure at the pipe outlet
ρ	Fluid density
ω	Shaft angular velocity
r_B	Radius of the pipe outlet

Connections A and B are conserving hydraulic ports associated with the pipe inlet and outlet, respectively. Connection W is the physical signal input port that provides the shaft angular velocity value.

Basic Assumptions and Limitations

- The pipe is assumed to be short, to be simulated with the fixed orifice.
- Flow is assumed to be fully developed along the pipe length.
- The transition between laminar and turbulent regimes is assumed to be sharp and taking place exactly at $Re=Re_{cr}$.
- Fluid inertia is not taken into account.
- Fluid compressibility is not taken into account.

Parameters

Pipe cross-section area

Cross-sectional area of the pipe. The default value is `1e-4` m^2.

Port B radius of rotation

Radius of rotation of the outer end of the pipe. The default value is `0.03` m.

Flow discharge coefficient

Semi-empirical parameter for pipe capacity characterization. Its value depends on the geometrical properties of the pipe, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.6`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches this value. The value of the parameter depends on the pipe geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Examples

For an example on page 1-334 of using the Rotating Pipe block to model a custom single-acting hydraulic cylinder, see the Centrifugal Force in Rotating Cylinder block reference page.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the pipe inlet.

`B`

Hydraulic conserving port associated with the pipe outlet.

`W`

Physical signal port providing the angular velocity value.

See Also

Centrifugal Force in Rotating Cylinder

Introduced in R2010a

Segmented Pipeline

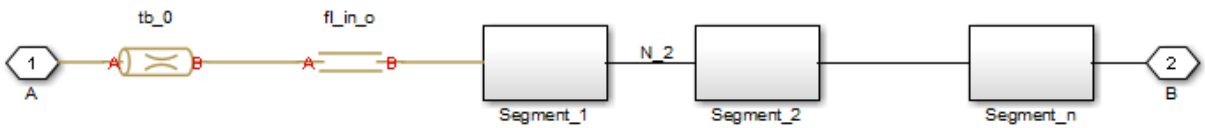
Hydraulic pipeline with resistive, fluid inertia, and fluid compressibility properties

Library

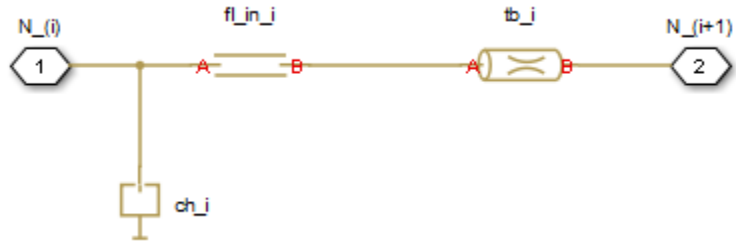
Pipelines

Description

The Segmented Pipeline block models hydraulic pipelines with circular cross sections. Hydraulic pipelines, which are inherently distributed parameter elements, are represented with sets of identical, connected in series, lumped parameter segments. It is assumed that the larger the number of segments, the closer the lumped parameter model becomes to its distributed parameter counterpart. The equivalent circuit of a pipeline adopted in the block is shown below, along with the segment configuration.



Pipeline Equivalent Circuit



Segment Configuration

The model contains as many Constant Volume Hydraulic Chamber blocks as there are segments. The chamber lumps fluid volume equal to

$$V = \frac{\pi \cdot d^2}{4} \frac{L}{N}$$

where

V	Fluid volume
d	Pipe diameter
L	Pipe length
N	Number of segments

The Constant Volume Hydraulic Chamber block is placed between two branches, each consisting of a Hydraulic Resistive Tube block and a Fluid Inertia block. Every Hydraulic Resistive Tube block lumps $(L+L_ad)/(N+1)$-th portion of the pipe length, while Fluid Inertia block has $L/(N+1)$ length (L_ad denotes additional pipe length equal to aggregate equivalent length of pipe local resistances, such as fitting, elbows, bends, and so on).

The nodes to which Constant Volume Hydraulic Chamber blocks are connected are assigned names `N_1`, `N_2`, ..., `N_`*n* (*n* is the number of segments). Pressures at these nodes are assumed to be equal to average pressure of the segment. Intermediate nodes between Hydraulic Resistive Tube and Fluid Inertia blocks are assigned names `nn_0`, `nn_1`, `nn_2`, ..., `nn_`*n*. The Constant Volume Hydraulic Chamber blocks are named `ch_1`, `ch_2`, ..., `ch_`*n*, Hydraulic Resistive Tube blocks are named `tb_0`, `tb_1`, `tb_2`, ..., `tb_`*n*, and Fluid Inertia blocks are named `fl_in_0`, `fl_in_1`, `fl_in_2`, ..., `fl_in_`*n*.

The number of segments determines the number of computational nodes associated with the block. A higher number increases model fidelity but decreases simulation speed. Experiment with different numbers to obtain a suitable trade-off between accuracy and speed. Use the following equation as a starting point in estimating a suitable number of segments:

$$N > \frac{4L}{\pi \cdot c}\omega$$

where:

N	Number of segments
L	Pipe length
c	Speed of sound in the fluid
ω	Maximum frequency to be observed in the pipe response

The table contains an example of simulation of a pipeline where the first four true eigenfrequencies are 89.1 Hz, 267 Hz, 446 Hz, and 624 Hz.

Number of Segments	1st Mode	2nd Mode	3rd Mode	4th Mode
1	112.3	–	–	–
2	107.2	271.8	–	–
4	97.7	284.4	432.9	689
8	93.2	271.9	435.5	628

The error between simulated and actual eigenfrequencies is less than 5% if an eight-segment model is used.

The flow rate through the pipeline is positive if it is directed from port A to port B. The pressure differential is positive if pressure is higher at port A than at port B.

Basic Assumptions and Limitations

Flow is assumed to be fully developed along the pipe length.

Parameters

Pipe internal diameter

Internal diameter of the pipe. The default value is `0.01` m.

Pipe length

Pipe geometrical length. The default value is `5` m.

Number of segments

Number of lumped parameter segments in the pipeline model. The default value is `1`.

Aggregate equivalent length of local resistances

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. This length is added to the geometrical pipe length only for hydraulic resistance computation. Both the fluid volume and fluid inertia are determined based on pipe geometrical length only. The default value is `1` m.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Pipe wall type

The parameter can have one of two values: `Rigid` or `Flexible`. If the parameter is set to `Rigid`, wall compliance is not taken into account, which can improve computational efficiency. The value `Flexible` is recommended for hoses and metal pipes where wall compliance can affect the system behavior. The default value is `Rigid`.

Static pressure-diameter coefficient

Coefficient that establishes relationship between the pressure and the internal diameter at steady-state conditions. This coefficient can be determined analytically for cylindrical metal pipes or experimentally for hoses. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`, and the default value is `2e-10` m/Pa.

Viscoelastic process time constant

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The default value is `0.008` s.

Specific heat ratio

Gas-specific heat ratio for the Constant Volume Hydraulic Chamber block. The default value is `1.4`.

Initial pressures at model nodes

Lets you specify the initial condition for pressure inside the pipe segments. The parameter can have one of two values:

- `The same initial pressure for all nodes` — The initial pressure in all pipe segments is the same, and is specified by the **Initial pressure** parameter value. This is the default.
- `Custom` — Lets you specify initial pressure individually for each pipe segment, by using the **Initial pressure vector** parameter. The vector size must be equal to the number of pipe segments, defined by the **Number of segments** parameter value.

Initial pressure

Specifies the initial pressure in all pipe segments. The parameter is used if the **Initial pressures at model nodes** parameter is set to `The same initial pressure for all nodes`, and the default value is `0`.

Initial pressure vector

Lets you specify initial pressure individually for each pipe segment. The parameter is used if the **Initial pressures at model nodes** parameter is set to `Custom`. The vector size must be equal to the number of pipe segments, defined by the **Number of segments** parameter value.

Initial flow rate

Specifies the initial volumetric flow rate through the pipe. The default value is `0 m^3/s`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Pipe wall type**
- **Initial pressures at model nodes**

All other block parameters are available for modification.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the pipe inlet.

B

Hydraulic conserving port associated with the pipe outlet.

See Also

Hydraulic Pipeline | Hydraulic Resistive Tube | Linear Hydraulic Resistance

Introduced in R2006a

Segmented Pipe LP

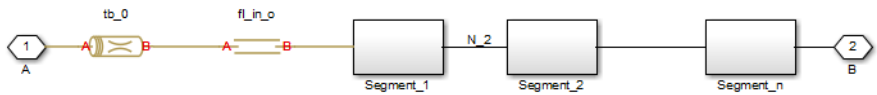
Hydraulic pipeline with resistive, fluid inertia, fluid compressibility, and elevation properties

Library

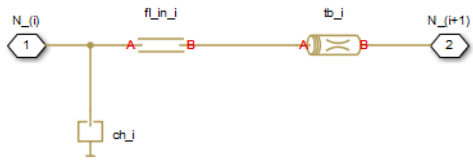
Low-Pressure Blocks

Description

The Segmented Pipe LP block models hydraulic pipelines with circular cross sections. Hydraulic pipelines, which are inherently distributed parameter elements, are represented with sets of identical, connected in series, lumped parameter segments. It is assumed that the larger the number of segments, the closer the lumped parameter model becomes to its distributed parameter counterpart. The equivalent circuit of a pipeline adopted in the block is shown below, along with the segment configuration.



Pipeline Equivalent Circuit



Segment Configuration

The model contains as many Constant Volume Hydraulic Chamber blocks as there are segments. The chamber lumps fluid volume equal to

$$V = \frac{\pi \cdot d^2}{4} \frac{L}{N}$$

where

V	Fluid volume
d	Pipe diameter
L	Pipe length
N	Number of segments

The Constant Volume Hydraulic Chamber block is placed between two branches, each consisting of a Resistive Pipe LP block and a Fluid Inertia block. Every Resistive Pipe LP block lumps `(L+L_ad)/(N+1)`-th portion of the pipe length, while Fluid Inertia block has `L/(N+1)` length (`L_ad` denotes additional pipe length equal to aggregate equivalent length of pipe local resistances, such as fitting, elbows, bends, and so on).

The nodes to which Constant Volume Hydraulic Chamber blocks are connected are assigned names `N_1`, `N_2`, ..., `N_n` (*n* is the number of segments). Pressures at these nodes are assumed to be equal to average pressure of the segment. Intermediate nodes between Resistive Pipe LP and Fluid Inertia blocks are assigned names `nn_0`, `nn_1`, `nn_2`, ..., `nn_n`. The Constant Volume Hydraulic Chamber blocks are named `ch_1`, `ch_2`, ..., `ch_n`, Resistive Pipe LP blocks are named `tb_0`, `tb_1`, `tb_2`, ..., `tb_n`, and Fluid Inertia blocks are named `fl_in_0`, `fl_in_1`, `fl_in_2`, ..., `fl_in_n`.

The number of segments determines the number of computational nodes associated with the block. A higher number increases model fidelity but decreases simulation speed. Experiment with different numbers to obtain a suitable trade-off between accuracy and speed. Use the following equation as a starting point in estimating a suitable number of segments:

$$N > \frac{4L}{\pi \cdot c} \omega$$

where

N	Number of segments
L	Pipe length
c	Speed of sound in the fluid
ω	Maximum frequency to be observed in the pipe response

The table below contains an example of simulation of a pipeline where the first four true eigenfrequencies are 89.1 Hz, 267 Hz, 446 Hz, and 624 Hz.

Number of Segments	1st Mode	2nd Mode	3rd Mode	4th Mode
1	112.3	–	–	–
2	107.2	271.8	–	–
4	97.7	284.4	432.9	689
8	93.2	271.9	435.5	628

As you can see, the error is less than 5% if an eight-segmented version is used.

The difference in elevation between ports A and B is distributed evenly between pipe segments.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B, and the pressure loss is determined as $\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}}$, .

Basic Assumptions and Limitations

Flow is assumed to be fully developed along the pipe length.

Parameters

- “Basic Parameters Tab” on page 1-1094
- “Wall Compliance Tab” on page 1-1095
- “Vertical Position Tab” on page 1-1095

Basic Parameters Tab

Pipe internal diameter

Internal diameter of the pipe. The default value is `0.01` m.

Pipe length

Pipe geometrical length. The default value is `5` m.

Number of segments

Number of lumped parameter segments in the pipeline model. The default value is `1`.

Aggregate equivalent length of local resistances

This parameter represents total equivalent length of all local resistances associated with the pipe. You can account for the pressure loss caused by local resistances, such as bends, fittings, armature, inlet/outlet losses, and so on, by adding to the pipe geometrical length an aggregate equivalent length of all the local resistances. This length is added to the geometrical pipe length only for hydraulic resistance computation. Both the fluid volume and fluid inertia are determined based on pipe geometrical length only. The default value is `1` m.

Internal surface roughness height

Roughness height on the pipe internal surface. The parameter is typically provided in data sheets or manufacturer's catalogs. The default value is `1.5e-5` m, which corresponds to drawn tubing.

Laminar flow upper margin

Specifies the Reynolds number at which the laminar flow regime is assumed to start converting into turbulent. Mathematically, this is the maximum Reynolds number at fully developed laminar flow. The default value is `2000`.

Turbulent flow lower margin

Specifies the Reynolds number at which the turbulent flow regime is assumed to be fully developed. Mathematically, this is the minimum Reynolds number at turbulent flow. The default value is `4000`.

Initial liquid pressure

Gauge pressure in the pipe segments at time zero. Enter a scalar for a single-segment pipeline and a vector for a multi-segment pipeline. The number of elements in the vector must match the number of segments in the pipe. The default value is `0` Pa.

Wall Compliance Tab

Pipe wall type

The parameter can have one of two values: `Rigid` or `Flexible`. If the parameter is set to `Rigid`, wall compliance is not taken into account, which can improve computational efficiency. The value `Flexible` is recommended for hoses and metal pipes where wall compliance can affect the system behavior. The default value is `Rigid`.

Static pressure-diameter coefficient

Coefficient that establishes relationship between the pressure and the internal diameter at steady-state conditions. This coefficient can be determined analytically for cylindrical metal pipes or experimentally for hoses. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `2e-12` m/Pa.

Viscoelastic process time constant

Time constant in the transfer function that relates pipe internal diameter to pressure variations. By using this parameter, the simulated elastic or viscoelastic process is approximated with the first-order lag. The value is determined experimentally or provided by the manufacturer. The parameter is used if the **Pipe wall type** parameter is set to `Flexible`. The default value is `0.01` s.

Specific heat ratio

Gas-specific heat ratio for the Constant Volume Hydraulic Chamber block. The default value is `1.4`.

Vertical Position Tab

Port A elevation wrt reference plane

Vertical position of port **A** with respect to a reference plane. The reference plane is assumed to be the same as that used in the **Port B elevation from reference plane** parameter. The default value is `0`m.

Port B elevation wrt reference plane

Vertical position of port **B** with respect to a reference plane. The reference plane is assumed to be the same as that used in the **Port A elevation from reference plane** parameter. The default value is `0`m.

Gravitational acceleration

Value of the gravitational acceleration constant (g). The block uses this parameter to compute the effects of an elevation gradient between the ports on their pressure differential. The default value is `9.80655` m/s^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Pipe wall type**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Pipe wall type** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the pipe inlet.

`B`

Hydraulic conserving port associated with the pipe outlet.

References

[1] White, F.M., *Viscous Fluid Flow*, McGraw-Hill, 1991

See Also

Hydraulic Pipe LP | Hydraulic Pipeline | Hydraulic Resistive Tube | Linear Hydraulic Resistance | Resistive Pipe LP | Segmented Pipeline

Introduced in R2009a

Shuttle Valve

Hydraulic valve that allows flow in one direction only

Library

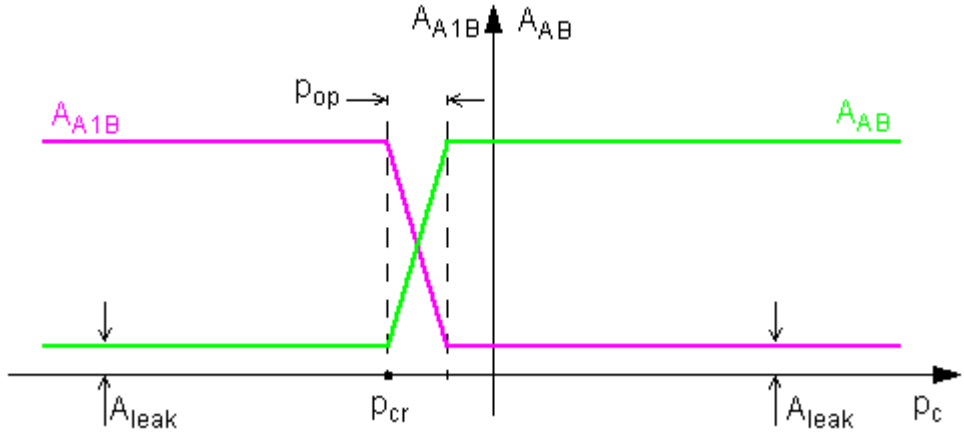
Directional Valves

Description

The Shuttle Valve block represents a hydraulic shuttle valve as a data-sheet-based model. The valve has two inlet ports (A and A1) and one outlet port (B). The valve is controlled by pressure differential $p_c = p_A - p_{A1}$. The valve permits flow either between ports A and B or between ports A1 and B, depending on the pressure differential p_c. Initially, path A-B is assumed to be opened. To open path A1-B (and close A-B at the same time), pressure differential must be less than the valve cracking pressure (p_{cr} <=0).

When cracking pressure is reached, the valve control member (spool, ball, poppet, etc.) is forced off its seat and moves to the opposite seat, thus opening one passage and closing the other. If the flow rate is high enough and pressure continues to change, the control member continues to move until it reaches its extreme position. At this moment, one of the valve passage areas is at its maximum. The valve maximum area and the cracking and maximum pressures are generally provided in the catalogs and are the three key parameters of the block.

The relationship between the A-B, A1–B path openings and control pressure p_c is shown in the following illustration.



In addition to the maximum area, the leakage area is also required to characterize the valve. The main purpose of the parameter is not to account for possible leakage, even though this is also important, but to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation. Therefore, the parameter value must be greater than zero.

The flow rate through each of the orifices is determined according to the following equations:

$$q_{AB} = C_D \cdot A_{AB} \sqrt{\frac{2}{\rho}} \cdot \frac{p_{AB}}{\left(p_{AB}^2 + p_{cr}^2\right)^{1/4}}$$

$$q_{A1B} = C_D \cdot A_{A1B} \sqrt{\frac{2}{\rho}} \cdot \frac{p_{A1B}}{\left(p_{A1B}^2 + p_{cr}^2\right)^{1/4}}$$

$$A_{AB} = \begin{cases} A_{leak} & \text{for } p_c \le p_{crack} \\ A_{leak} + k \cdot (p_c - p_{crack}) & \text{for } p_{crack} < p_c < p_{crack} + p_{op} \\ A_{\max} & \text{for } p_c \ge p_{crack} + p_{op} \end{cases}$$

$$A_{A1B} = \begin{cases} A_{leak} & \text{for } p_c \ge p_{crack} + p_{op} \\ A_{\max} - k \cdot (p_c - p_{crack}) & \text{for } p_{crack} < p_c < p_{crack} + p_{op} \\ A_{\max} & \text{for } p_c \le p_{crack} \end{cases}$$

$$k = \frac{A_{\max} - A_{leak}}{p_{op}}$$

$$p_c = p_A - p_{A1}$$

where

q_{AB}, q_{A1B}	Flow rates through the AB and A1B orifices
p_{AB}, p_{A1B}	Pressure differentials across the AB and A1B orifices
p_A, p_{A1}, p_B	Gauge pressures at the block terminals
C_D	Flow discharge coefficient
ρ	Fluid density
A_{AB}, A_{A1B}	Instantaneous orifice AB and A1B passage areas
A_{max}	Fully open orifice passage area
A_{leak}	Closed valve leakage area
p_c	Valve control pressure differential
p_{crack}	Valve cracking pressure differential
p_{op}	Pressure differential needed to fully shift the valve
p_{crAB}, p_{crA1B}	Minimum pressures for turbulent flow across the AB and A1B orifices

The minimum pressures for turbulent flow across the AB and A1B orifices, p_{crAB} and p_{crA1B}, are calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{crAB} = (p_{avgAB} + p_{atm})(1 - B_{lam})$$

$$p_{crA1B} = (p_{avgA1B} + p_{atm})(1 - B_{lam})$$

$$p_{avgAB} = (p_A + p_B)/2$$

$$p_{avgA1B} = (p_{A1} + p_B)/2$$

where

p_{avgAB}	Average pressure for orifice AB
p_{avgA1B}	Average pressure for orifice A1B
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{crAB} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_{HAB}}\right)^2$$

$$p_{crA1B} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_{HA1B}}\right)^2$$

$$D_{HAB} = \sqrt{\frac{4A_{AB}}{\pi}}$$

$$D_{HA1B} = \sqrt{\frac{4A_{A1B}}{\pi}}$$

where

D_{HAB}, D_{HA1B}	Instantaneous orifice hydraulic diameters
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

By default, the block does not include valve opening dynamics. Adding valve opening dynamics provides continuous behavior that is particularly helpful in situations with rapid valve opening and closing. The orifice passage areas A_{AB} and A_{A1B} in the equations above then become steady-state orifice AB and A1B passage areas, respectively. Instantaneous orifice AB and A1B passage areas with opening dynamics are determined as follows:

$$A_{AB_dyn}(t=0) = A_{AB_init}$$

$$\frac{dA_{AB_dyn}}{dt} = \frac{A_{AB} - A_{AB_dyn}}{\tau}$$

$$A_{A1B_dyn} = A_{\max} + A_{leak} - A_{AB_dyn}$$

where

A_{AB_dyn}	Instantaneous orifice AB passage area with opening dynamics
A_{A1B_dyn}	Instantaneous orifice A1B passage area with opening dynamics
A_{AB_init}	Initial open area for orifice AB
τ	Time constant for the first order response of the valve opening
t	Time

The block positive direction is from port A to port B and from port A1 to port B. Control pressure is determined as $p_c = p_A - p_{A1}$.

Basic Assumptions and Limitations

- Valve opening is linearly proportional to the pressure differential.
- No loading on the valve, such as inertia, friction, spring, and so on, is considered.

Parameters

Maximum passage area

Valve passage maximum cross-sectional area. The default value is `1e-4` m^2.

Cracking pressure

Pressure differential level at which the orifice of the valve starts to open. The default value is `-1e4` Pa.

Opening pressure

Pressure differential across the valve needed to shift the valve from one extreme position to another. The default value is `1e4` Pa.

Flow discharge coefficient

Semi-empirical parameter for valve capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`, which corresponds to a round orifice in thin material with sharp edges. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed valve. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Opening dynamics

Select one of the following options:

- `Do not include valve opening dynamics` — The valve sets its orifice passage area directly as a function of pressure. If the area changes instantaneously, so does the flow equation. This is the default.
- `Include valve opening dynamics` — Provide continuous behavior that is more physically realistic, by adding a first-order lag during valve opening and closing. Use this option in hydraulic simulations with the local solver for real-time simulation. This option is also helpful if you are interested in valve opening dynamics in variable step simulations.

Opening time constant

The time constant for the first order response of the valve opening. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `0.1` s.

Initial area at port A

The initial open area for orifice AB. This parameter is available only if **Opening dynamics** is set to `Include valve opening dynamics`. The default value is `1e-4` m^2.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Opening dynamics**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Opening dynamics** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the valve inlet.

`A1`

Hydraulic conserving port associated with the valve inlet.

`B`

Hydraulic conserving port associated with the valve outlet.

See Also

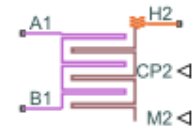
Check Valve | Pilot-Operated Check Valve

Introduced in R2006b

Simple Heat Exchanger (G)

Simple model of a heat exchanger between a gas network and a general fluid

Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers



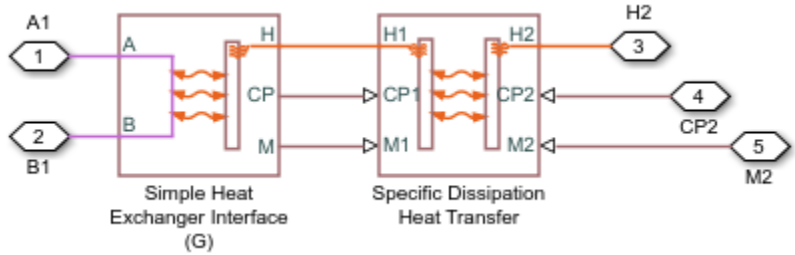
Description

The Simple Heat Exchanger (G) block models the heat transfer and fluid dynamics of a heat exchanger located between a gas network and an external fluid. The model is based on tabulated data relating the operating conditions inside the heat exchanger—the mass flow rate, pressure drop, and specific dissipation factor, a measure of heat exchange effectiveness.

The fluid properties of the gas network do not apply to the external fluid. These are specified for the gas network through a Gas Properties (G) block and for the external fluid through physical signals. Port **CP2** provides the isobaric specific heat of the external fluid and port **M2** provides its mass flow rate. The rate of heat transfer is computed using the specific dissipation method.

Component Structure

This block is a composite component assembled from simpler components found in the **Fundamental Components** library. A Specific Dissipation Heat Transfer block captures the heat transfer between the gas networks. A Simple Heat Exchanger Interface (G) block captures the pressure drop and temperature change between the gas inlets.



Component Block Diagram

Ports

Input

`CP2` — Isobaric specific heat of the controlled fluid

physical signal

Isobaric specific heat of the controlled fluid.

`M2` — Entrance mass flow rate of the controlled fluid

physical signal

Entrance mass flow rate of the controlled fluid. Positive values indicate flow into the heat exchanger. Negative values indicate flow out of the heat exchanger.

Conserving

`A1` — Gas inlet

gas

Opening through which gas 1 can enter and exit the heat exchanger.

`B1` — Gas inlet

gas

Opening through which gas can enter and exit the heat exchanger.

`H2` — Inlet temperature of external fluid
thermal

Thermal port associated with the inlet temperature of the external fluid.

Parameters

Heat Transfer Tab

`Gas mass flow rate vector, mdot1` — Mass flow rate at which to specify specific-dissipation data
`[.3, .5, .6, .7, 1, 1.4, 1.9, 2.3] kg/s` (default) | *M*-element array with units of mass/time

Array of mass flow rates at the gas inlet. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Controlled fluid mass flow rate vector, mdot2` — Mass flow rate at which to specify specific-dissipation data
`[.3, .5, 1, 1.3, 1.7, 2, 2.6, 3.3] kg/s` (default) | *N*-element array with units of mass/time

Array of mass flow rates at the inlet for the controlled fluid. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Specific dissipation table, SD(mdot1, mdot2)` — Specific dissipation values corresponding to specified mass flow rates
8-by-8 matrix with units of `kW/K` (default) | *M*-by-*N* matrix with units of power/temperature

Matrix of specific dissipation values corresponding to the specified mass flow rate arrays for the gas and controlled fluid. The block uses the tabulated data to calculate the heat transfer at the simulated operating conditions.

`Check if violating maximum specific dissipation` — Warning condition for out-of-bounds specific dissipation
`Warning` (default) | `None`

Option to warn if the specific dissipation falls outside the bounds of the specified tabulated data.

Pressure Loss Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data
`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates
`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet corresponding to the tabulated mass flow rate data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified
`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified
`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-6 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Gas volume` — Volume of gas inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of gas occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A1 and B1` — Flow area at the gas inlets
`0.01 m^2` (default) | scalar with units of area

Flow area at the gas inlets. Inlets **A1** and **B1** are assumed to be identical in size.

Effects and Initial Conditions Tab

`Gas initial temperature` — Temperature inside the heat exchanger at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of gas at the start of simulation.

`Gas initial pressure` — Pressure inside the heat exchanger at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of gas at the start of simulation.

See Also

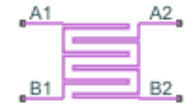
Simple Heat Exchanger Interface (G) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger (G-G)

Simple model of a heat exchanger between two gas networks

Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers



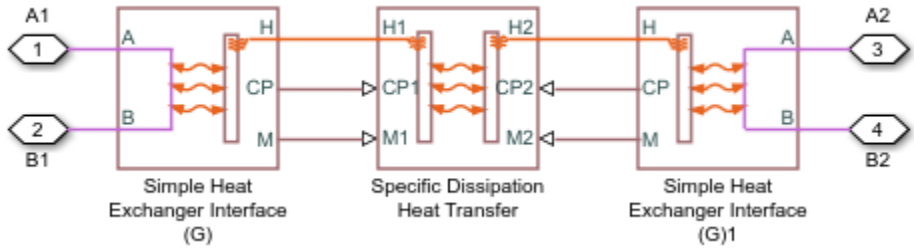
Description

The Simple Heat Exchanger (G-G) block models the heat transfer and fluid dynamics of a heat exchanger located between two gas networks. The model is based on tabulated data relating the operating conditions inside the heat exchanger—the mass flow rate, pressure drop, and specific dissipation factor, a measure of heat exchange effectiveness.

The gas networks are considered distinct. Each network must connect to a separate fluid properties block—Gas Properties (G) in each case. The fluids do not mix nor do they change phase. Their only interaction is heat transfer across a common boundary. The rate of heat transfer is computed using the specific dissipation method.

Component Structure

This block is a composite component assembled from simpler components found in the **Fundamental Components** library. A Specific Dissipation Heat Transfer block captures the heat transfer between the gas networks. Two Simple Heat Exchanger Interface (G) blocks capture the pressure drop and temperature change between the gas inlets.



Component Block Diagram

Ports

Conserving

A1 — Gas 1 inlet

gas

Opening through which gas 1 can enter and exit the heat exchanger.

B1 — Gas 1 inlet

gas

Opening through which gas 1 can enter and exit the heat exchanger.

A2 — Gas 2 inlet

gas

Opening through which gas 2 can enter and exit the heat exchanger.

B2 — Gas 2 inlet

gas

Opening through which gas 2 can enter and exit the heat exchanger.

Parameters

Heat Transfer Tab

`Gas mass flow rate vector, mdot1` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, .6, .7, 1, 1.4, 1.9, 2.3] kg/s` (default) | *M*-element array with units of mass/time

Array of mass flow rates at the inlet for gas 1. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Gas mass flow rate vector, mdot2` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, 1, 1.3, 1.7, 2, 2.6, 3.3] kg/s` (default) | *N*-element array with units of mass/time

Array of mass flow rates at the inlet for gas 2. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Specific dissipation table, SD(mdot1, mdot2)` — Specific dissipation values corresponding to specified mass flow rates

8-by-8 matrix with units of `kW/K` (default) | *M*-by-*N* matrix with units of power/temperature

Matrix of specific dissipation values corresponding to the specified mass flow rate arrays for the two gases. The block uses the tabulated data to calculate the heat transfer at the simulated operating conditions.

`Check if violating maximum specific dissipation` — Warning condition for out-of-bounds specific dissipation

`Warning` (default) | `None`

Option to warn if the specific dissipation falls outside the bounds of the specified tabulated data.

Gas 1|2 Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data
`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates
`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet corresponding to the tabulated mass flow rate data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified
`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified
`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-6 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Gas 1|2 volume` — Volume of gas inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of gas occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A1|A2 and B1|B2` — Flow area at the gas inlets

`0.01 m^2` (default) | scalar with units of area

Flow area at the gas inlets. Inlets **A** and **B** are assumed to be identical in size for each gas.

Effects and Initial Conditions Tab

`Gas 1|2 initial temperature` — Temperature inside the heat exchanger at the start of simulation

`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of gas at the start of simulation.

`Gas 1|2 initial pressure` — Pressure inside the heat exchanger at the start of simulation

`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of gas at the start of simulation.

See Also

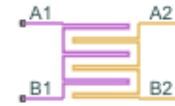
Simple Heat Exchanger Interface (G) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger (G-TL)

Simple model of a heat exchanger between a gas circuit and a thermal liquid network

Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers



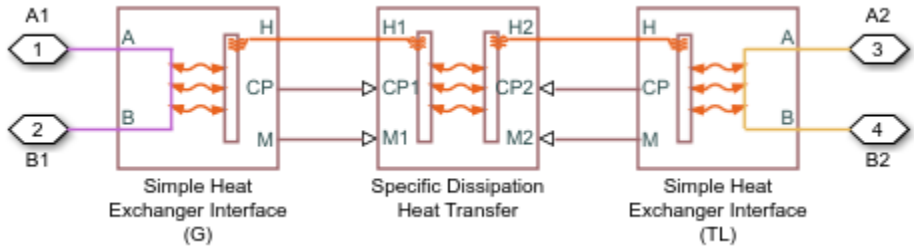
Description

The Simple Heat Exchanger (G-TL) block models the heat transfer and fluid dynamics of a heat exchanger located between a gas network and a thermal liquid network. The model is based on tabulated data relating the operating conditions inside the heat exchanger—the mass flow rate, pressure drop, and specific dissipation factor, a measure of heat exchange effectiveness.

The gas and thermal liquid networks are considered distinct. Each network must connect to a separate fluid properties block—Gas Properties (G) or Thermal Liquid Settings (TL). The fluids do not mix nor do they change phase. Their only interaction is heat transfer across a common boundary. The rate of heat transfer is computed using the specific dissipation method.

Component Structure

This block is a composite component assembled from simpler components found in the **Fundamental Components** library. A Specific Dissipation Heat Transfer block captures the heat transfer between the gas and thermal liquid networks. A Simple Heat Exchanger Interface (G) block and a Simple Heat Exchanger Interface (TL) block capture the pressure drop and temperature change between the inlets.



Component Block Diagram

Ports

Conserving

A1 — Gas inlet

gas

Opening through which gas can enter and exit the heat exchanger.

B1 — Gas inlet

gas

Opening through which gas can enter and exit the heat exchanger.

A2 — Thermal liquid inlet

thermal liquid

Opening through which thermal liquid can enter and exit the heat exchanger.

B2 — Thermal liquid inlet

thermal liquid

Opening through which thermal liquid can enter and exit the heat exchanger.

Parameters

Heat Transfer Tab

`Gas 1 mass flow rate vector, mdot1` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, .6, .7, 1, 1.4, 1.9, 2.3] kg/s` (default) | *M*-element array with units of mass/time

Array of mass flow rates at the gas inlet. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Thermal Liquid 2 mass flow rate vector, mdot2` — Mass flow rate at which to specify specific-dissipation data

`[.3, .5, 1, 1.3, 1.7, 2, 2.6, 3.3] kg/s` (default) | *N*-element array with units of mass/time

Array of mass flow rates at the thermal liquid inlet. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Specific dissipation table, SD(mdot1, mdot2)` — Specific dissipation values corresponding to specified mass flow rates

8-by-8 matrix with units of `kW/K` (default) | *M*-by-*N* matrix with units of power/temperature

Matrix of specific dissipation values corresponding to the specified mass flow rate arrays for the gas and controlled fluid. The block uses the tabulated data to calculate the heat transfer at the simulated operating conditions.

`Check if violating maximum specific dissipation` — Warning condition for out-of-bounds specific dissipation

`Warning` (default) | `None`

Option to warn if the specific dissipation falls outside the bounds of the specified tabulated data.

Gas 1 Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data
`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates
`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet corresponding to the tabulated mass flow rate data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified
`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified
`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-6 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Gas 1 volume` — Volume of gas inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of gas occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A1 and B1` — Flow area at the gas inlets
`0.01 m^2` (default) | scalar with units of area

Flow area at the gas inlets. Inlets **A1** and **B1** are assumed to be identical in size.

Thermal Liquid 2 Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data
`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates
`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet corresponding to the tabulated mass flow rate data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified
`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified
`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-3 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Thermal Liquid 2 volume` — Volume of thermal liquid inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of thermal liquid occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A2and B2` — Flow area at the thermal liquid inlets
`0.01 m^2` (default) | scalar with units of area

Flow area at the thermal liquid inlets. Inlets **A2** and **B2** are assumed to be identical in size.

Effects and Initial Conditions Tab

`Gas 1 initial temperature` — Temperature inside the heat exchanger at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of gas at the start of simulation.

`Gas 1 initial pressure` — Pressure inside the heat exchanger at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of gas at the start of simulation.

`Thermal Liquid 2 dynamic compressibility` — Option to model pressure dynamics inside the heat exchanger
`On` (default) | `Off`

Option to model the pressure dynamics inside the heat exchanger. Setting this parameter to `Off` removes the pressure derivative terms from the component energy and mass conservation equations. The pressure inside the heat exchanger is then reduced to the weighted average of the two port pressures.

`Thermal Liquid 2 initial temperature` — Temperature inside the heat exchanger at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of thermal liquid at the start of simulation.

`Thermal Liquid 2 initial pressure` — Pressure inside the heat exchanger at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of thermal liquid at the start of simulation.

See Also

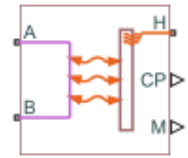
Simple Heat Exchanger Interface (G) | Simple Heat Exchanger Interface (TL) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger Interface (G)

Thermal interface between a gas and its surroundings

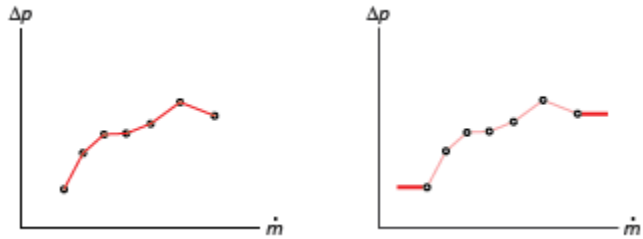
Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers / Fundamental Components



Description

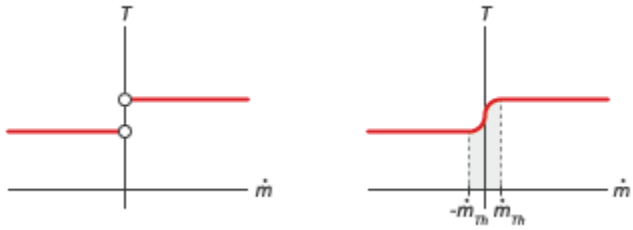
The Simple Heat Exchanger Interface (G) block models the pressure drop and temperature change in a gas as it traverses the length of a thermal interface such as that provided by a heat exchanger. Heat transfer across the thermal interface is ignored. See the composite block diagram of the Simple Heat Exchanger (G-G) block for an example showing how to combine the two blocks.

The pressure drop is calculated as a function of mass flow rate from tabulated data specified at some reference pressure and temperature. The calculation is based on linear interpolation if the mass flow rate is within the bounds of the tabulated data and on nearest-neighbor extrapolation otherwise. In other words, neighboring data points connect through straight-line segments, with those at the mass flow rate bounds extending horizontally outward.



Linear interpolation (left) and nearest-neighbor extrapolation (right)

The block calculations rely on the states and properties of the fluid—temperature, density, and specific internal energy—at the entrance to the thermal interface. The entrance changes abruptly from one port to the other during flow reversal, introducing discontinuities in the values of these variables. To eliminate these discontinuities, the block smooths the affected variables at mass flow rates below a specified threshold value.



Smoothing of entrance temperature below mass flow rate threshold

Mass Balance

Mass can enter and exit the thermal interface through ports **A** and**B**. The volume of the interface is fixed but the compressibility of the fluid means that the mass inside the interface can change with pressure and temperature. The compressibility of the gas is always taken into account, with its value being that specified in the Gas Properties (G) block dialog box. The mass balance in the interface can then be expressed as:

$$\frac{\partial M}{\partial p}\frac{dp}{dt}+\frac{\partial M}{\partial T}\frac{dT}{dt}=\dot{m}_{\mathrm{A}}+\dot{m}_{\mathrm{B}},$$

where:

- M is the mass of the internal fluid volume of the thermal interface.
- p is the internal fluid pressure.
- T is the internal fluid temperature.
- $\dot{m}_{*}$ are the mass flow rates in through the gas ports.

Energy Balance

Energy can enter and exit the thermal interface in two ways: with fluid flow through ports **A** and **B** and with heat flow through port **H**. No work is done on or by the fluid inside the interface. The rate of energy accumulation in the internal fluid volume of the interface must therefore equal the sum of the energy flow rates through all three ports:

$$\frac{\partial E}{\partial p}\frac{dp}{dt}+\frac{\partial E}{\partial T}\frac{dT}{dt}=\phi_{\mathrm{A}}+\phi_{\mathrm{B}}+Q_{\mathrm{H}},$$

where:

- E is the total energy in the internal fluid volume of the thermal interface.
- ϕ_* are the energy flow rates in through the gas ports.
- Q is the heat flow rate in through the thermal port.

Momentum Balance

The pressure drop calculation is based entirely on tabulated data that you specify. The causes of the pressure drop are ignored, except in the effects that they might have on the specified data. The overall pressure drop from one gas port to the other is calculated from the individual pressure drops from each gas port to the internal fluid volume:

$$p_{\mathrm{A}}-p_{\mathrm{B}}=\Delta p_{\mathrm{A}}-\Delta p_{\mathrm{B}},$$

where:

- p_* are the fluid pressures at the gas ports.
- Δp_* are the pressure drops from the gas ports to the internal fluid volume:

  $$\Delta p_*=p_*-p,$$

  with p as the pressure in the internal fluid volume.

The tabulated data is specified at a reference pressure and temperature from which a third reference parameter, the *reference density*, is calculated. The ratio of the reference density to the actual port density serves as a correction factor in the individual pressure drop equations, each defined as:

$$\Delta p_* = \Delta p(\dot{m}_*)\frac{\rho_R}{\rho_*},$$

where:

- $\Delta p(\dot{m})$ is the tabulated pressure drop function.
- ρ_* are the fluid densities at the gas ports.

The asterisk denotes the gas port (**A** or **B**) at which a parameter or variable is defined. Subscript R denotes a reference value. The density at the interface entrance is smoothed below the mass flow rate threshold by introducing a hyperbolic term α:

$$\rho_{*,\text{smooth}} = \rho_*\left(\frac{1+\alpha}{2}\right) + \rho\left(\frac{1-\alpha}{2}\right),$$

where ρ_{smooth} is the smoothed density at the entrance port, ρ_* is the unsmoothed density at the same port, and ρ is the density in the internal fluid volume. The hyperbolic smoothing term is defined as:

$$\alpha = \tanh\left(4\frac{\dot{m}_{\text{avg}}}{\dot{m}_{\text{th}}}\right),$$

where $\dot{m}_{avg}$ is the average of the mass flow rates through the gas ports and $\dot{m}_{th}$ is the mass flow rate threshold specified in the block dialog box. This threshold determines the width of the mass flow rate region over which to smooth the fluid density. The average mass flow rate is defined as:

$$\dot{m}_{\text{avg}} = \frac{\dot{m}_A + \dot{m}_B}{2}$$

Ports

Output

`CP` — Isobaric specific heat of gas, `kJ/(kg*K)`

physical signal

Isobaric specific heat of the gas in the internal fluid volume of the thermal interface.

`M` — Mass flow rate of the thermal liquid, kg/s
physical signal

Mass flow rate of the gas in the internal fluid volume of the interface. The output signal is positive when the flow rate is directed from port **A** to port **B** and negative otherwise.

Conserving

`A` — Fluid opening
gas

Opening through which gas can enter and exit the thermal interface.

`B` — Fluid opening
gas

Opening through which gas can enter and exit the thermal interface.

`H` — Thermal condition at the fluid entrance
thermal

Thermal port used to set the thermal condition at the gas port serving as the gas entrance.

Parameters

Pressure Loss Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data
`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates
`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet corresponding to the tabulated mass flow rate data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified
`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified
`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified. The block uses this parameter to calculate a third reference parameter, the reference density. The reference that it uses to scale the tabulated pressure drop data for pressures and temperatures deviating from the reference conditions.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-6 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Gas volume` — Volume of gas inside the heat exchanger
`0.01 m^3` (default) | scalar with units of volume

Volume of gas occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A1 and B1` — Flow area at the gas inlets
`0.01 m^2` (default) | scalar with units of area

Flow area at the gas inlets. Inlets **A1** and **B1** are assumed to be identical in size.

Variables Tab

`Temperature of gas volume` — Temperature of the internal gas volume at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of gas at the start of simulation.

`Pressure of gas volume` — Pressure of the internal gas volume at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of gas at the start of simulation.

`Density of gas volume` — Density of the internal gas volume at the start of simulation
`1.2 kg/m^3` (default) | scalar with units of mass/volume

Density of the internal volume of gas at the start of simulation.

See Also

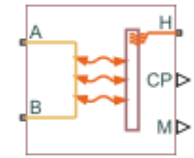
Simple Heat Exchanger (G) | Simple Heat Exchanger Interface (TL) | Specific Dissipation Heat Transfer

Introduced in R2017b

Simple Heat Exchanger Interface (TL)

Thermal interface between a thermal liquid and its surroundings

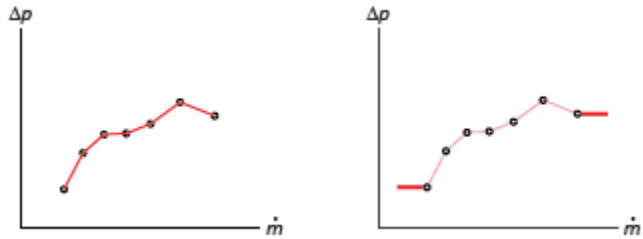
Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers / Fundamental Components



Description

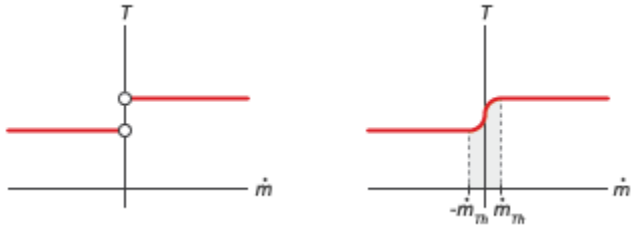
The Simple Heat Exchanger Interface (TL) block models the pressure drop and temperature change in a thermal liquid as it traverses the length of a thermal interface such as that provided by a heat exchanger. Heat transfer across the thermal interface is ignored. See the composite block diagram of the Simple Heat Exchanger (TL-TL) block for an example showing how to combine the two blocks.

The pressure drop is calculated as a function of mass flow rate from tabulated data specified at some reference pressure and temperature. The calculation is based on linear interpolation if the mass flow rate is within the bounds of the tabulated data and on nearest-neighbor extrapolation otherwise. In other words, neighboring data points connect through straight-line segments, with those at the mass flow rate bounds extending horizontally outward.



Linear interpolation (left) and nearest-neighbor extrapolation (right)

The block calculations rely on the states and properties of the fluid—temperature, density, and specific internal energy—at the entrance to the thermal interface. The entrance changes abruptly from one port to the other during flow reversal, introducing discontinuities in the values of these variables. To eliminate these discontinuities, the block smooths the affected variables at mass flow rates below a specified threshold value.



Smoothing of entrance temperature below mass flow rate threshold

Mass Balance

Mass can enter and exit the thermal interface through ports **A** and**B**. The volume of the interface is fixed but the compressibility of the fluid means that the mass inside the interface can change with pressure and temperature. Whether compressibility is factored into the block calculations depends on the setting of the **Thermal Liquid dynamic compressibility** parameter in the **Effects and Initial Conditions** tab:

$$\dot{m}_A + \dot{m}_B = \begin{cases} \left(\frac{dp}{dt}\frac{1}{\beta} - \frac{dT}{dt}\alpha \right)\rho V, & \text{if } \textbf{Thermal Liquid dynamic compressibility} \text{ is On} \\ 0, & \text{otherwise} \end{cases},$$

where:

- $\dot{m}_*$ are the mass flow rates in through the thermal liquid ports.
- p is the internal fluid pressure.
- T is the internal fluid temperature.
- α is the isobaric thermal expansion coefficient.

- β is the isothermal bulk modulus.
- ρ is the internal fluid density.
- V is the internal fluid volume.

If the **Thermal Liquid dynamic compressibility** parameter is set to `Off`, the fluid is treated as incompressible and the mass flow rate in through one thermal liquid port must exactly equal that out through the other thermal liquid port. The rate of mass accumulation is, in this case, zero.

Energy Balance

Energy can enter and exit the thermal interface in two ways: with fluid flow through ports **A** and **B** and with heat flow through port **H**. No work is done on or by the fluid inside the interface. The rate of energy accumulation in the internal fluid volume of the interface must therefore equal the sum of the energy flow rates through all three ports:

$$\frac{\partial E}{\partial p}\frac{dp}{dt}+\frac{\partial E}{\partial T}\frac{dT}{dt}=\phi_{\mathrm{A}}+\phi_{\mathrm{B}}+Q_{\mathrm{H}},$$

where:

- E is the total energy in the internal fluid volume of the thermal interface.
- ϕ_* are the energy flow rates in through the thermal liquid ports.
- Q is the heat flow rate in through the thermal port.

Momentum Balance

The pressure drop calculation is based entirely on tabulated data that you specify. The causes of the pressure drop are ignored, except in the effects that they might have on the specified data. The overall pressure drop from one thermal liquid port to the other is calculated from the individual pressure drops from each thermal liquid port to the internal fluid volume:

$$p_{\mathrm{A}}-p_{\mathrm{B}}=\Delta p_{\mathrm{A}}-\Delta p_{\mathrm{B}},$$

where:

- p_* are the fluid pressures at the thermal liquid ports.

- Δp_* are the pressure drops from the thermal liquid ports to the internal fluid volume:

  $$\Delta p_* = p_* - p,$$

  with p as the pressure in the internal fluid volume.

The tabulated data is specified at a reference pressure and temperature from which a third reference parameter, the *reference density*, is calculated. The ratio of the reference density to the actual port density serves as a correction factor in the individual pressure drop equations, each defined as:

$$\Delta p_* = \Delta p(\dot{m}_*)\frac{\rho_{\mathrm{R}}}{\rho_*},$$

where:

- $\Delta p(\dot{m})$ is the tabulated pressure drop function.
- ρ_* are the fluid densities at the thermal liquid ports.

The asterisk denotes the thermal liquid port (**A** or **B**) at which a parameter or variable is defined. Subscript R denotes a reference value. The density at the interface entrance is smoothed below the mass flow rate threshold by introducing a hyperbolic term α:

$$\rho_{*,\mathrm{smooth}} = \rho_*\left(\frac{1+\alpha}{2}\right) + \rho\left(\frac{1-\alpha}{2}\right),$$

where ρ_{smooth} is the smoothed density at the entrance port, ρ_* is the unsmoothed density at the same port, and ρ is the density in the internal fluid volume. The hyperbolic smoothing term is defined as:

$$\alpha = \tanh\left(4\frac{\dot{m}_{\mathrm{avg}}}{\dot{m}_{\mathrm{th}}}\right),$$

where $\dot{m}_{avg}$ is the average of the mass flow rates through the thermal liquid ports and $\dot{m}_{th}$ is the mass flow rate threshold specified in the block dialog box. This threshold determines the width of the mass flow rate region over which to smooth the fluid density. The average mass flow rate is defined as:

$$\dot{m}_{\text{avg}} = \frac{\dot{m}_{\text{A}} + \dot{m}_{\text{B}}}{2}$$

Ports

Output

`CP` — Isobaric specific heat of the thermal liquid, `kJ/(kg*K)`

physical signal

Isobaric specific heat of the thermal liquid in the internal fluid volume of the thermal interface.

`M` — Mass flow rate of the thermal liquid, `kg/s`

physical signal

Mass flow rate of the thermal liquid in the interior of the thermal interface. The output signal is positive when the flow rate is directed from port **A** to port **B** and negative otherwise.

Conserving

`A` — Fluid opening

thermal liquid

Opening through which thermal liquid can enter and exit the thermal interface.

`B` — Fluid opening

thermal liquid

Opening through which thermal liquid can enter and exit the thermal interface.

`H` — Thermal condition at the fluid entrance

thermal

Thermal port used to set the thermal condition at the thermal liquid port serving as the thermal liquid entrance.

Parameters

Pressure Loss Tab

`Mass flow rate vector` — Mass flow rates at which to specify pressure-drop data

`[.3, .5, 1, 1.5, 2, 2.5] kg/s` (default) | *K*-element array with units of mass/time

Array of mass flow rates at which to specify the pressure drop tabulated data.

`Pressure drop vector` — Pressure-drop data corresponding to specified mass flow rates

`[3, 5, 10, 25, 35, 50] * 1e-3 MPa` (default) | *K*-element array with units of pressure

Array of pressure drops from inlet to outlet corresponding to the tabulated mass flow rate data.

`Reference inflow temperature` — Temperature at which pressure-drop data is specified

`293.15 K` (default) | scalar with units of temperature

Temperature at which the tabulated pressure-drop data is specified.

`Reference inflow pressure` — Pressure at which pressure-drop tabulated data is specified

`0.101325 MPa` (default) | scalar with units of pressure

Pressure at which the tabulated pressure-drop data is specified. The block uses this parameter to calculate a third reference parameter, the reference density. The reference that it uses to scale the tabulated pressure drop data for pressures and temperatures deviating from the reference conditions.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data

`1e-6 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Thermal Liquid volume` — Volume of thermal liquid inside the heat exchanger

`0.01 m^3` (default) | scalar with units of volume

Volume of thermal liquid occupying the heat exchanger at any given time. The initial conditions specified in the **Effects and Initial Conditions** tab apply to this volume. The volume is constant during simulation.

`Cross-sectional area at ports A1 and B1` — Flow area at the thermal liquid inlets
`0.01 m^2` (default) | scalar with units of area

Flow area at the thermal liquid inlets. Inlets **A1** and **B1** are assumed to be identical in size.

Effects and Initial Conditions

`Thermal Liquid dynamic compressibility` — Option to model pressure dynamics inside the heat exchanger
`On` (default) | `Off`

Option to model the pressure dynamics inside the heat exchanger. Setting this parameter to `Off` removes the pressure derivative terms from the component energy and mass conservation equations. The pressure inside the heat exchanger is then reduced to the weighted average of the two port pressures.

`Thermal Liquid initial temperature` — Temperature inside the heat exchanger at the start of simulation
`293.15 K` (default) | scalar with units of temperature

Temperature of the internal volume of thermal liquid at the start of simulation.

`Thermal Liquid initial pressure` — Pressure inside the heat exchanger at the start of simulation
`0.101325 MPa` (default) | scalar with units of pressure

Pressure of the internal volume of thermal liquid at the start of simulation.

See Also

Heat Exchanger (TL) | Heat Exchanger Interface (TL) | Simple Heat Exchanger Interface (G) | Specific Dissipation Heat Transfer

Introduced in R2017b

Single-Acting Actuator (TL)

Linear actuator with piston motion controlled by one thermal liquid chamber

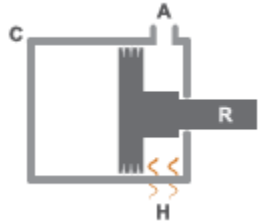
Library

Thermal Liquid/Actuators

Description

The Single-Acting Actuator (TL) block models a linear actuator with piston motion controlled by a single thermal liquid chamber. The actuator generates force in the extension and retraction strokes, but the actuation force depends on the gauge pressure at a single chamber.

The figure shows the key components of the actuator model. Port A represents the thermal liquid chamber inlet. Port R represents the translating actuator piston and port C the actuator case. Port H represents the thermal interface between the thermal liquid chamber and the environment.



Single-Acting Actuator Schematic

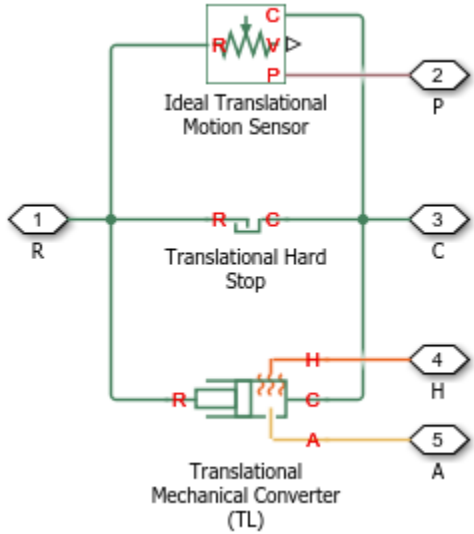
The direction of the piston motion depends on the mechanical orientation setting in the block dialog box. If the mechanical orientation is positive, then a positive gauge pressure at port A yields a positive piston translation relative to the actuator case. The direction of motion reverses for a negative mechanical orientation.

A set of hard stops limit the piston range of motion. The hard stops are treated as spring-damper systems. The spring stiffness coefficient controls the restorative component of the hard-stop contact force and the damping coefficient the dissipative component.

The hard stops are located at the distal ends of the piston stroke. If the mechanical orientation is positive, then the lower hard stop is at $x = 0$ and the upper hard stop at $x = +stroke$. If the mechanical orientation is negative, then the lower hard stop is at $x = -stroke$ and the upper hard stop at $x = 0$.

This block is a composite component based on the Simscape Foundation blocks:

- Translational Mechanical Converter (TL)
- Translational Hard Stop
- Ideal Translational Motion Sensor



Composite Component Diagram

Parameters

Actuator Tab

Mechanical orientation

Orientation of the actuator piston relative to the direction of flow. A positive orientation causes the piston to move in the positive direction relative to the actuator casing in response to a positive flow rate through port A. The default setting is `Positive`.

The mechanical orientation affects the placement of the actuator hard stops. One hard stop is always at position zero. The second hard stop is at the piston stroke distance if the mechanical orientation is positive and at minus the piston stroke distance if the mechanical orientation is negative.

Piston cross-sectional area

Area normal to the direction of flow in actuator chamber A. The block uses this area to calculate the hydraulic force due to the fluid pressure in chamber A. The piston cross-sectional area must be greater than zero. The default value is `0.01` m^2.

Piston stroke

Maximum distance the actuator piston can travel. The piston stroke must be greater than zero. The default value is `0.1` m.

Hard stops limit piston motion to the length of the piston stroke. One hard stop is located at position zero. The second hard stop is at the piston stroke distance if **Mechanical Orientation** is set to `Positive` and at minus the piston stroke if **Mechanical Orientation** is set to `Negative`.

Dead volume

Fluid volume remaining in the actuator chamber at a zero piston displacement. The block uses this volume to account for mass and energy storage in the chamber when the piston is at position zero. The dead volume must be greater than zero. The default value is `1e-5` m^3.

Environment pressure specification

Choice of environment pressure. Options include `Atmospheric pressure` and `Specified pressure`. Selecting `Specified pressure` exposes an additional parameter, **Environment pressure**.

Environment pressure

Pressure outside the actuator casing. This pressure acts against the pressures inside the actuator chamber. A value of zero corresponds to a vacuum. The default value is `0.101325` MPa. This parameter is visible only when **Environment pressure specification** is set to `Specified pressure`.

Hard Stop Tab

Hard-stop stiffness coefficient

Spring coefficient of the actuator hard stops. The spring coefficient accounts for the restorative portion of the hard-stop contact force. Increase the coefficient value to model harder contact. The default value is `1e10` N/m.

Hard-stop damping coefficient

Damping coefficient of the actuator hard stops. The damping coefficient accounts for the dissipative portion of the hard-stop contact force. Increase the coefficient value to reduce bounce upon contact. The default value is `150` N/(m/s).

Hard stop model

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` (default) — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.
- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

Transition region

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero. The default value is `1 mm`..

Initial Conditions Tab

Piston initial displacement

Piston position at the start of simulation. This value must be between zero and the piston stroke if the **Mechanical orientation** parameter is set to `Positive`. It must be between zero and minus the piston stroke if the **Mechanical orientation** parameter is set to `Negative`. The default value is `0` m.

Initial liquid temperature

Temperature in actuator chamber A at the start of simulation. The default value is `293.15` K.

Fluid dynamic compressibility

Option to model effects due to fluid dynamic compressibility. Select `On` to enable fluid dynamic compressibility and `Off` to disable it.

Initial liquid pressure

Pressure in actuator chamber A at the start of simulation. The default value is `0.101325` MPa.

Ports

- A — Thermal liquid conserving port representing chamber A
- C — Translational conserving port representing the actuator casing
- R — Translational conserving port representing the actuator piston
- H — Thermal conserving port representing the thermal interface between chamber A and the environment
- P — Physical signal output port for the piston position data

See Also

Double-Acting Actuator (TL)

Introduced in R2016a

Single-Acting Hydraulic Cylinder

Hydraulic actuator exerting force in one direction

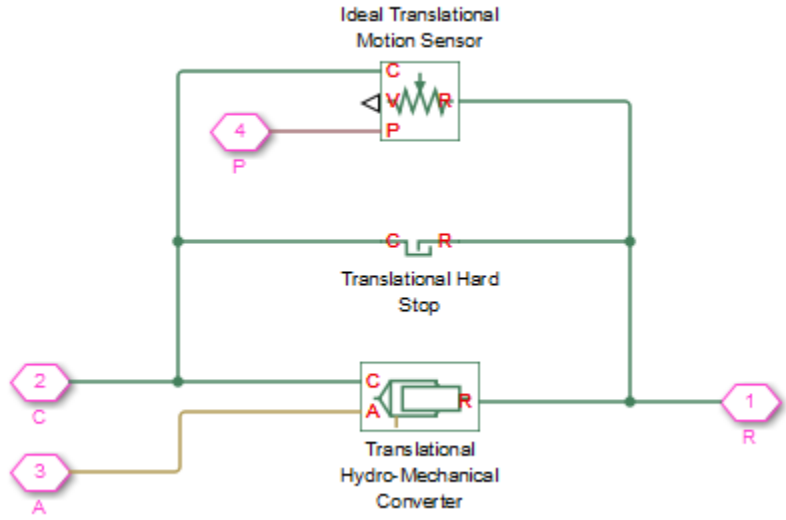
Library

Hydraulic Cylinders

Description

The Single-Acting Hydraulic Cylinder block models a device that converts hydraulic energy into mechanical energy in the form of translational motion. Hydraulic fluid pumped under pressure into the cylinder chamber forces the piston to move and exert force on the cylinder rod. Single-acting cylinders transfer force and motion in one direction only. Use an external device, such as a spring, weight, or another opposite installed cylinder, to move the rod in the opposite direction.

The model of the cylinder is built of Simscape Foundation library blocks. The schematic diagram of the model is shown below.



Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connection A is a hydraulic conserving port associated with the cylinder inlet. The physical signal output port P provides rod displacement, calculated as follows:

$$x_{pst} = x_0 + x_p$$

where

x_{pst}	Rod displacement output by the physical signal port
x_0	Initial distance between piston and cap
x_p	Rod displacement with respect to its initial position

The energy through port A is directed to the Translational Hydro-Mechanical Converter block. The converter transforms hydraulic energy into mechanical energy and accounts for the fluid compressibility in the cylinder chamber. The rod motion is limited with the mechanical Translational Hard Stop block in such a way that the rod can travel only between cylinder caps. The piston position is conveyed to the physical signal port P.

The block directionality is adjustable and can be controlled with the **Cylinder orientation** parameter.

Basic Assumptions and Limitations

- No leakage, internal or external, is taken into account.
- No loading on piston rod, such as inertia, friction, spring, and so on, is taken into account. If necessary, you can easily add them by connecting an appropriate building block to cylinder port R.

Parameters

Piston area

Effective piston area. The default value is `0.001` m^2.

Piston stroke

Piston maximum travel between caps. The default value is `0.1` m.

Initial distance between piston and cap

The distance between the piston and cap at the beginning of simulation. This value cannot exceed the piston stroke. The default value is `0`, which corresponds to the fully retracted position.

Dead volume

Fluid volume that remains in the chamber after the rod is fully retracted. The default value is `1e-4` m^3.

Chamber initial pressure

Pressure in the cylinder chamber at the beginning of simulation. The default value is `0`.

Specific heat ratio

Gas-specific heat ratio for the Hydraulic Piston Chamber block. The default value is `1.4`.

Contact stiffness

Specifies the elastic property of colliding bodies for the Translational Hard Stop block. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is `1e6` N/m.

Contact damping

Specifies dissipating property of colliding bodies for the Translational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater

the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N*s/m.

Hard stop model

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` (default) — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.
- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

Transition region

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero. The default value is `1 mm`..

Cylinder orientation

Specifies cylinder orientation with respect to the globally assigned positive direction. The cylinder can be installed in two different ways, depending upon whether it exerts force in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port A exerts force in negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Cylinder orientation**

All other block parameters are available for modification.

Global Parameters

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the cylinder inlet.

`R`

Mechanical translational conserving port associated with the cylinder rod.

`C`

Mechanical translational conserving port associated with the cylinder clamping structure.

`P`

Physical signal output port that outputs rod extension.

See Also

Double-Acting Hydraulic Cylinder | Ideal Translational Motion Sensor | Single-Acting Hydraulic Cylinder (Simple) | Translational Hard Stop | Translational Hydro-Mechanical Converter

Introduced in R2006a

Single-Acting Hydraulic Cylinder (Simple)

Basic functionality of single-acting hydraulic cylinder

Library

Hydraulic Cylinders

Description

The Single-Acting Hydraulic Cylinder (Simple) block represents a simplified version of a hydraulic cylinder, developed for applications where only the basic cylinder functionality must be reproduced, in exchange for better numerical efficiency. For these reasons, such factors as fluid compressibility, friction, and leakages are assumed to be negligible. The hard stops are assumed to be fully inelastic, to eliminate any possible oscillations at the end of the stroke. The model is especially suitable for real-time and HIL (hardware-in-the-loop) simulation, if such simplifications are acceptable.

The model is described with the following equations:

$$F = A \cdot p - F_c$$

$$q = A \cdot v$$

$$\frac{dx}{dt} = v$$

$$v = v_R - v_C$$

$$F_c = \begin{cases} (x - x_E) \cdot K_p \cdot v & \text{if } x > x_E, v > 0 \\ (x - x_R) \cdot K_p \cdot v & \text{if } x < x_R, v < 0 \\ 0 & \text{otherwise} \end{cases}$$

$$x_E = S - x_0$$

$$x_R = -x_0$$

where

F	Force developed by the cylinder
v	Cylinder rod velocity
v_R, v_C	Absolute velocities of cylinder rod and cylinder case, respectively
A	Piston area
p	Pressure at the cylinder inlet
q	Flow rate at the cylinder inlet
x	Piston position
x_0	Initial distance between piston and the cap
F_c	Hard stop force
x_E	Distance the piston can travel to fully extend from initial position
x_R	Distance the piston can travel to fully retract from initial position
K_p	Penetration coefficient
S	Piston stroke

The main difference between this block and the Single-Acting Hydraulic Cylinder block is the representation of the hard stop:

- The Single-Acting Hydraulic Cylinder block utilizes the elastic model of the stop, which is essentially a combination of a spring and a damper, connected in parallel. The stiffness of the spring, which represents contact stiffness of colliding bodies, is very high, resulting in high-frequency, low-magnitude oscillations at the moment the bodies collide. These oscillations can hardly be noticed experimentally, but they are difficult to process by a numerical simulation and generally decrease numerical efficiency.

- The hard stop in the Single-Acting Hydraulic Cylinder (Simple) block is represented with the inelastic model, which is essentially a viscous damper with the penetration-dependent damping coefficient. This coefficient is referred to as the penetration coefficient. With inelastic model, no oscillation is generated during an impact, resulting in improved numerical robustness and efficiency. But inelastic stop model has a feature that you must consider while selecting the model: colliding bodies continue slowly moving into each other as long as the contact is loaded with the compressing force. In real life, this phenomenon is similar to collision of two bodies separated by a sizable layer of viscous liquid. It takes some time to squeeze the liquid before bodies themselves come into contact.

Connections R and C are mechanical translational conserving ports corresponding to the cylinder rod and cylinder clamping structure, respectively. Connection A is a hydraulic conserving port associated with the cylinder inlet. The physical signal output port P shows current rod extension.

Pressure at port A generates force in the direction specified by the **Cylinder orientation** parameter.

Basic Assumptions and Limitations

- Friction between moving parts is not taken into account.
- Inertia effects are not taken into account.
- Fluid compressibility is not taken into account.
- Leakage flows are assumed to be negligible.
- The hard stops are assumed to be fully inelastic, as explained above.

Parameters

Piston area

Effective piston area. The default value is `5e-4` m^2.

Piston stroke

Piston maximum travel between caps. The default value is `0.1` m.

Initial distance between piston and cap

The distance between the piston and cap at the beginning of simulation. This value cannot exceed the piston stroke. The default value is `0`, which corresponds to the fully retracted position.

Penetration coefficient

Specifies the penetration property of colliding bodies. The higher the value of the coefficient, the less the bodies penetrate into each other, but the sharper the increase of the impact force. To find out the value of the coefficient for a particular application, it is recommended to run the same model with different hard stop models (or with different cylinder models) until they show close results. The default value of the coefficient is `1e12` N/m/(m/s).

Cylinder orientation

Specifies cylinder orientation with respect to the globally assigned positive direction. The cylinder can be installed in two different ways, depending upon whether it exerts force in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port A exerts force in negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Cylinder orientation**

All other block parameters are available for modification.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the cylinder inlet.

`R`

Mechanical translational conserving port associated with the cylinder rod.

`C`

Mechanical translational conserving port associated with the cylinder clamping structure.

`P`

Physical signal output port that outputs rod extension.

See Also

Double-Acting Hydraulic Cylinder | Double-Acting Hydraulic Cylinder (Simple) | Single-Acting Hydraulic Cylinder | Translational Hard Stop | Translational Hydro-Mechanical Converter

Introduced in R2010a

Single-Acting Rotary Actuator

Single-acting hydraulic rotary actuator

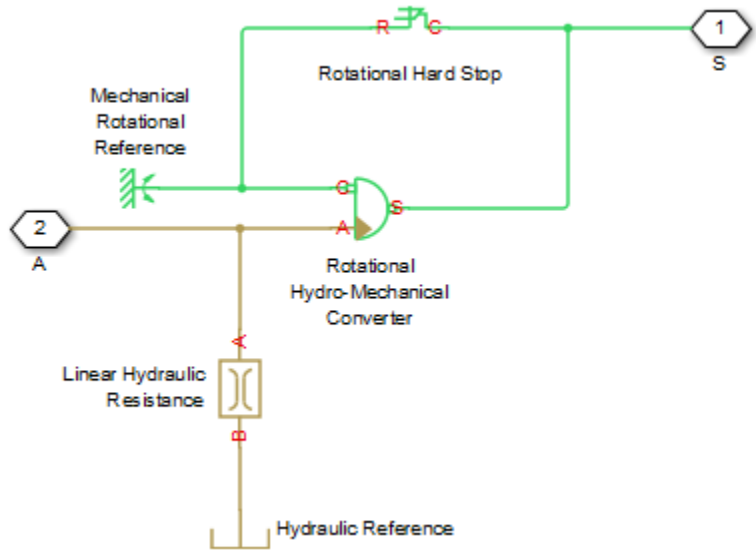
Library

Hydraulic Cylinders

Description

The Single-Acting Rotary Actuator block models a single-acting hydraulic rotary actuator, which directly converts hydraulic energy into mechanical rotational energy without employing intermediary transmissions such as rack-and-pinion, sliding spline, chain, and so on. Single-acting actuators generate torque and motion in a single direction only. Use an external device, such as a spring or another opposite installed actuator, to move the shaft in the opposite direction.

The model of the actuator is built of Simscape Foundation library blocks. The schematic diagram of the model is shown below.



The blocks in the diagram perform the following functions:

Rotational Hydro-Mechanical Converter	Converts hydraulic energy into mechanical rotational energy and vice versa, while accounting for fluid compressibility.
Rotational Hard Stop	Imposes limits on shaft rotation.
Linear Hydraulic Resistance	Accounts for leakages.

Connection A is a hydraulic conserving port corresponding to the actuator chamber. Connection S is a mechanical rotational conserving port associated with the actuator shaft.

The block directionality is adjustable and can be controlled with the **Actuator orientation** parameter.

Basic Assumptions and Limitations

No loading, such as inertia, friction, spring, and so on, is taken into account. If necessary, you can easily add them by connecting an appropriate building block to port S.

Parameters

Actuator displacement

Effective displacement of the actuator. The default value is `4.5e-5` m^3/rad.

Shaft stroke

Shaft maximum travel between stops. The default value is `5.1` rad.

Shaft initial angle

The position of the shaft at the beginning of simulation. You can set the shaft position to any angle within its stroke. The default value is `0`, which corresponds to the shaft position at the very beginning of the stroke.

Dead volume

Fluid volume that remains in the chamber when the shaft is positioned at the very beginning of the stroke. The default value is `1e-4` m^3.

Leak coefficient

Leak coefficient for the Linear Hydraulic Resistance block. The default value is `1e-14` (m^3/s)/Pa.

Specific heat ratio

Gas-specific heat ratio for the Hydraulic Piston Chamber block. The default value is `1.4`.

Contact stiffness

Specifies the elastic property of colliding bodies for the Rotational Hard Stop block. The greater the value of the parameter, the less the bodies penetrate into each other, the more rigid the impact becomes. Lesser value of the parameter makes contact softer, but generally improves convergence and computational efficiency. The default value is `1e6` N*m/rad.

Contact damping

Specifies dissipating property of colliding bodies for the Rotational Hard Stop block. At zero damping, the impact is close to an absolutely elastic one. The greater the value of the parameter, the more energy dissipates during an interaction. Keep in mind that damping affects slider motion as long as the slider is in contact with the stop, including the period when slider is pulled back from the contact. For computational efficiency and convergence reasons, MathWorks recommends that you assign a nonzero value to this parameter. The default value is 150 N*m/(rad/s).

Hard stop model

Modeling approach for hard stops. Options include:

- `Stiffness and damping applied smoothly through transition region` (default) — Scale the magnitude of the contact force from zero to its full value over a specified transition length. The scaling is polynomial in nature. The polynomial scaling function is numerically smooth and it produces no zero crossings of any kind.
- `Full stiffness and damping applied at bounds, undamped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during penetration and a spring force—without a damping component—during rebound. No smoothing is applied.
- `Full stiffness and damping applied at bounds, damped rebound` — Apply the full value of the calculated contact force when the hard-stop location is breached. The contact force is a mix of spring and damping forces during both penetration and rebound. No smoothing is applied. This is the hard-stop model used in previous releases.

Transition region

Distance below which scaling is applied to the hard-stop force. The contact force is zero when the distance to the hard stop is equal to the value specified here. It is at its full value when the distance to the hard stop is zero. The default value is `1 mm`..

Actuator orientation

Specifies actuator orientation with respect to the globally assigned positive direction. The actuator can be installed in two different ways, depending upon whether it generates torque in the positive or in the negative direction when pressure is applied at its inlet. If pressure applied at port A generates torque in the negative direction, set the parameter to `Acts in negative direction`. The default value is `Acts in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameter:

- **Actuator orientation**

All other block parameters are available for modification.

Global Parameters

Parameter determined by the type of working fluid:

- **Fluid bulk modulus**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the actuator inlet.

S

Mechanical rotational conserving port associated with the actuator shaft.

See Also

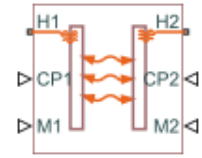
Linear Hydraulic Resistance | Rotational Hard Stop | Rotational Hydro-Mechanical Converter

Introduced in R2007a

Specific Dissipation Heat Transfer

Simple heat transfer model between two general fluids

Library: Simscape / Fluids / Fluid Network Interfaces / Heat Exchangers / Fundamental Components



Description

The Specific Dissipation Heat Transfer block models the heat transfer between two fluids given only minimal knowledge of component parameters. The fluids are controlled by physical signals, with these providing the entrance mass flow rate and isobaric specific heat for each. Thermal ports set the entrance temperatures of the fluids.

The rate of heat transfer is calculated from the specific dissipation, a parameter specified in tabulated form as a function of the entrance mass flow rates. The specific dissipation quantifies the amount of heat exchanged between the fluids per unit of time when the entrance temperatures differ by one degree.

Pressure losses and other aspects of flow mechanics are ignored. To capture such effects, use the heat exchanger interface blocks provided in the same library. Combine heat transfer and heat exchanger blocks to model a custom heat exchanger. See the composite block diagrams of the heat exchanger blocks for examples.

Heat Transfer Rate

Heat flows from the warmer fluid to the cooler fluid, at a rate proportional to the difference between the fluid entrance temperatures. The heat flow rate is positive if fluid 1 enters at a higher temperature than fluid 2—and therefore if heat flows from fluid 1 to fluid 2:

$$Q = SD\left(T_{1,\mathrm{in}} - T_{2,\mathrm{in}}\right),$$

where $T_{*,\text{in}}$ are the fluid entrance temperatures, determined by the conditions at thermal port **H1** for fluid 1 and **H2** for fluid 2. *SD* is the specific dissipation obtained from the specified tabulated data at the given mass flow rates:

$$SD = SD\left(\dot{m}_1, \dot{m}_2\right),$$

where $\dot{m}$ are the entrance mass flow rates, specified through physical signal port **M1** for fluid 1 and **M2** for fluid 2. The specific dissipation can be calculated for a set of entrance mass flow rates ($\dot{m}_1$, $\dot{m}_2$) given the experimental values of the heat transfer rate and the corresponding entrance temperature difference:

$$SD = \frac{Q}{T_{1,in} - T_{2,in}}.$$

The heat transfer rate is constrained so that the specific dissipation used in the calculations can never exceed the maximum value:

$$SD_{\max} = \min\left(C_1, C_2\right),$$

where C_* are the thermal capacity rates of the controlled fluids, each defined as:

$$C_* = \dot{m}_* c_{p,*},$$

with $c_{p,*}$ denoting the isobaric specific heat of the fluid, specified through physical signal port **CP1** for fluid 1 and **CP2** for fluid 2. The constraint on the maximum heat transfer rate is implemented in the form of a piecewise function:

$$Q = \begin{cases} SD_{max}\left(T_{1,in} - T_{2,in}\right), & \text{if } SD > SD_{max} \\ SD\left(T_{1,in} - T_{2,in}\right), & \text{otherwise} \end{cases},$$

A warning is issued whenever the heat flow rate exceeds the maximum value, $SD_{max}\left(T_{1,in} - T_{2,in}\right)$, if the **Check if violating maximum specific dissipation** block parameter is set to `Warning`.

Ports

Input

`CP1` — Isobaric specific heat of controlled fluid 1

physical signal

Isobaric specific heat of controlled fluid 1.

`CP2` — Isobaric specific heat of controlled fluid 2

physical signal

Isobaric specific heat of controlled fluid 2.

`M1` — Entrance mass flow rate of controlled fluid 1

physical signal

Entrance mass flow rate of controlled fluid 1. Positive values indicate flow into the heat exchanger. Negative values indicate flow out of the heat exchanger.

`M2` — Entrance mass flow rate of controlled fluid 2

physical signal

Entrance mass flow rate of controlled fluid 2. Positive values indicate flow into the heat exchanger. Negative values indicate flow out of the heat exchanger.

Conserving

`H1` — Entrance temperature of controlled fluid 1

thermal

Entrance temperature of controlled fluid 1.

`H2` — Entrance temperature of controlled fluid 2

thermal

Entrance temperature of controlled fluid 2.

Parameters

`Thermal Liquid mass flow rate vector, mdot1` — Entrance mass flow rate at which to specify specific-dissipation data
`[.3, .5, .6, .7, 1, 1.4, 1.9, 2.3] kg/s` (default) | *M*-element array with units of mass/time

Array of mass flow rates at the inlet for controlled fluid 1. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Controlled fluid mass flow rate vector, mdot2` — Entrance mass flow rate at which to specify specific-dissipation data
`[.3, .5, 1, 1.3, 1.7, 2, 2.6, 3.3] kg/s` (default) | *N*-element array with units of mass/time

Array of mass flow rates at the inlet for controlled fluid 2. Each value corresponds to a row in the specific dissipation lookup table. Positive values indicate flow into the heat exchanger and negative values indicate flow out of the heat exchanger.

`Specific dissipation table, SD(mdot1, mdot2)` — Specific dissipation values corresponding to specified mass flow rates
8-by-8 matrix with units of `kW/K` (default) | *M*-by-*N* matrix with units of power/temperature

Matrix of specific dissipation values corresponding to the specified mass flow rate arrays for controlled fluids 1 and 2. The block uses the tabulated data to calculate the heat transfer at the simulated operating conditions.

`Mass flow rate threshold for flow reversal` — Mass flow rate below which to smooth numerical data
`1e-3 kg/s` (default) | scalar with units of mass/time

Mass flow rate below which to initiate a smooth flow reversal to prevent discontinuities in the simulation data.

`Check if violating maximum specific dissipation` — Option to warn if the specific dissipation exceeds the maximum allowed value
`Warning` (default) | `None`

Option to warn if the specific dissipation exceeds the maximum value described in the block description.

See Also

Heat Exchanger Interface (TL)

Introduced in R2017b

Spool Orifice Hydraulic Force

Axial hydraulic force exerted on spool

Library

Valve Forces

Description

The Spool Orifice Hydraulic Force block simulates the steady-state axial hydraulic force exerted on the spool by fluid flowing through the orifice. The spool is assumed to be sharp-edged. You have two parameterization options:

- A rectangular slot, to be used with the Orifice with Variable Area Slot block
- A round hole, or a set of round holes, to be used with the Orifice with Variable Area Round Holes block

The width of the slot, or the diameter of the holes, must be considerably larger than the radial clearance between the spool and the sleeve. Connect the Spool Orifice Hydraulic Force block in series with the respective orifice block to monitor the flow rate.

The force is simulated according to the following equations:

$$F = \rho \frac{q^2}{A} \cos\theta \cdot or$$

$$\theta = 0.3663 + 0.8373(1 - \exp(-x / 1.848))$$

$$x = x_0 + s \cdot or$$

For rectangular slot,

$$A = \begin{cases} b \cdot \sqrt{x^2 + \delta^2} & \text{for } x > 0 \\ b \cdot \delta & \text{for } x <= 0 \end{cases}$$

For round holes,

$$A = \begin{cases} A_{\min} + Z\frac{d^2}{8}(\alpha - \sin\alpha) & \text{for d >= } x >= 0 \\ A_{\min} & \text{for } x < 0 \\ A_{\min} + A_{\max} & \text{for } x > d \end{cases}$$

$$\alpha = 2 \cdot \operatorname{acos}\left(1 - 2\frac{x}{d}\right)$$

$$A_{\min} = Z \cdot d \cdot \delta$$

$$A_{\max} = \pi d^2/4$$

where

F	Axial hydraulic force
q	Flow rate through the orifice
ρ	Fluid density
A	Orifice area
Θ	Jet angle (rad)
x_0	Orifice initial opening
s	Spool displacement
b	Orifice width, for rectangular slot
D	Orifice diameter, for round holes
Z	Number of round holes

δ	Radial clearance
or	Orientation parameter with respect to the globally assigned positive direction. If the orifice is opened while the spool is shifted in positive direction, *or* equals 1. If the orifice is opened while the spool is shifted in negative direction, *or* equals –1.

Connections A and B are hydraulic conserving ports that should be connected in series with the orifice block to monitor the flow rate. Connection S is a physical signal port that provides the spool displacement. Connection F is a physical signal port that outputs the hydraulic axial force value. This port should be connected to the control port of an Ideal Force Source block. The force computed in the block always acts to close the orifice.

Basic Assumptions and Limitations

- The transient effects are assumed to be negligible.
- The jet angle approximation is based on the Richard von Mises equation.
- The jet angle variation with the orifice opening is identical for the rectangular slot and the round holes orifices.
- The block can be used with orifices whose width or diameter is considerably larger than the axial opening.

Parameters

Orifice type

Select the type of orifice:

- `Rectangular slot` — Orifice created by the sharp-edged spool and a rectangular slot in the sleeve. This is the default.
- `Round holes` — Orifice created by the sharp-edged spool and a set of round holes evenly distributed along the sleeve perimeter.

Orifice width

Orifice width. The parameter must be greater than zero. The default value is `0.01` m. The parameter is used if **Orifice type** is set to `Rectangular slot`.

Diameter of round holes

Orifice diameter. If multiple orifices are used, all of them must be of the same diameter, identically aligned with respect to the spool edge, and spaced evenly among the sleeve perimeter. The parameter must be greater than zero. The default value is `0.005` m. The parameter is used if **Orifice type** is set to `Round holes`.

Number of round holes

Number of round holes in the sleeve. The parameter must be greater than zero. The default value is `4`. The parameter is used if **Orifice type** is set to `Round holes`.

Radial clearance

The radial clearance between the spool and the sleeve. The default value is `1e-5` m.

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is `0`.

Orifice orientation

The parameter is introduced to specify the effect of the force on the orifice opening. The parameter can be set to one of two options: `Opens in positive direction` or `Opens in negative direction`. The value `Opens in positive direction` specifies an orifice that opens when the spool moves in the globally assigned positive direction. The default value is `Opens in positive direction`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Orifice type**
- **Orifice orientation**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Orifice type** parameter at the time the model entered Restricted mode.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with an orifice inlet.

B

Hydraulic conserving port associated with an orifice outlet.

S

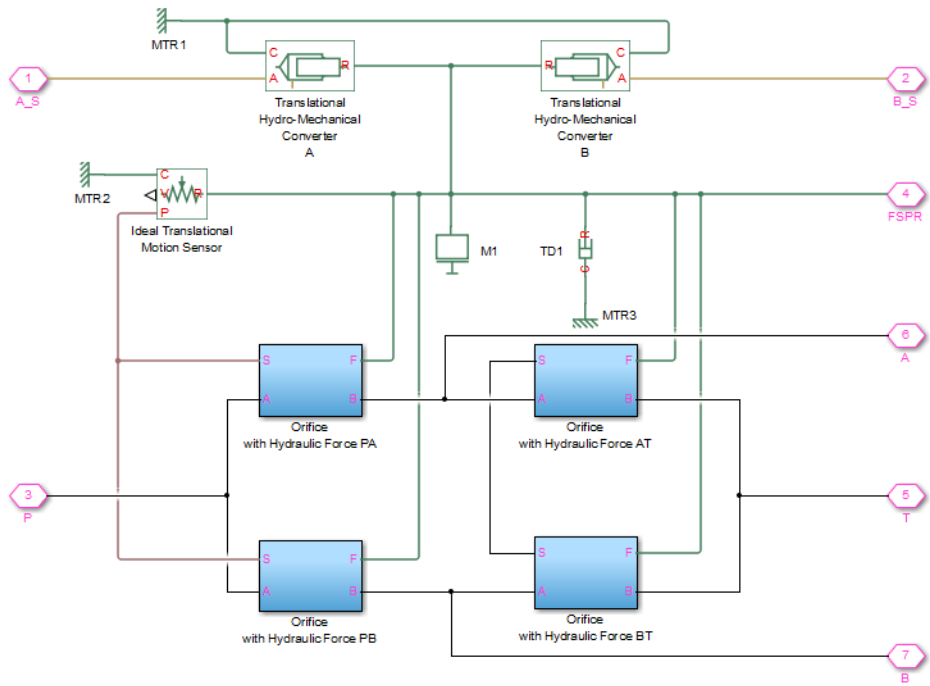
Physical signal port that provides the spool displacement.

F

Physical signal port that outputs hydraulic axial force.

Examples

The following example shows a model of a 4-way, 3-position, hydraulically-operated directional valve where the hydraulic axial forces acting on the spool are being taken into consideration.

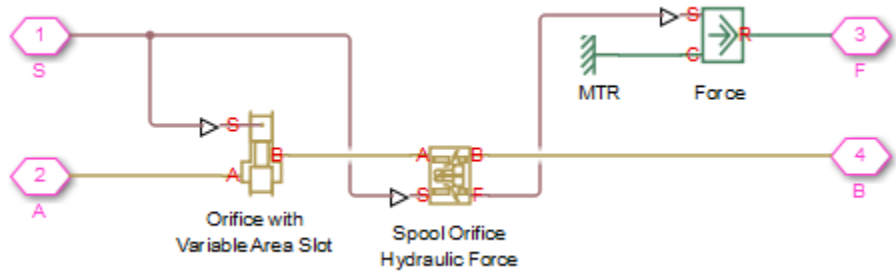


The spool (mass `M1`, viscous friction `TD1`) is shifted by the servo-actuator simulated by two Translational Hydro-Mechanical Converter blocks. Connections A_S and B_S are hydraulic ports for applying pilot control pressure.

Four variable orifices are represented by subsystems:

- Orifice with Hydraulic Force PA
- Orifice with Hydraulic Force PB
- Orifice with Hydraulic Force AT
- Orifice with Hydraulic Force BT

The structure of a subsystem is shown in the following illustration.



It consists of an Orifice with Variable Area Slot block, which simulates hydraulic properties of the orifice, connected in series with a Spool Orifice Hydraulic Force block. The force value computed in the block is exported through its port F and passed to the Force block.

The forces on all four orifices (`F_PA`, `F_PB`, `F_AT`, `F_BT`) are applied to the valve spool as it is shown in the first schematic.

For more details and for parameter settings, see the Servo-Valve Controlling Hydraulic Actuator example.

See Also

Valve Hydraulic Force

Introduced in R2007a

Spring-Loaded Accumulator

Hydraulic accumulator with spring used for energy storage

Library

Accumulators

Description

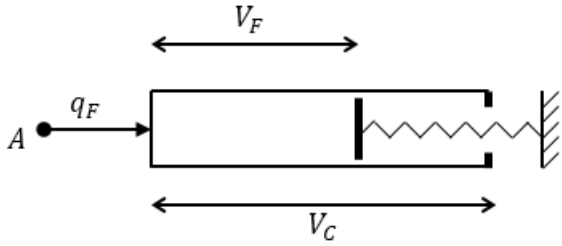
This block models a spring-loaded hydraulic fluid accumulator. The accumulator consists of a preloaded spring and a fluid chamber. The fluid chamber is connected to a hydraulic system.

As the fluid pressure at the accumulator inlet becomes greater than the preload pressure, fluid enters the accumulator and compresses the spring, storing hydraulic energy. A decrease in the fluid pressure causes the spring to decompress and discharge stored fluid into the system.

During typical operations, the spring pressure is equal to the pressure in the fluid chamber. However, if the pressure at the accumulator inlet drops below the preload pressure, the spring becomes isolated from the system. In this situation, the fluid chamber is empty and the spring pressure remains constant and equal to the preload pressure while the pressure at the accumulator inlet depends on the hydraulic system to which the accumulator is connected. If the pressure at the accumulator inlet builds up to the preload pressure or higher, fluid enters the accumulator again.

The motion of the spring is restricted by two hard stops that limit the expansion and contraction of the fluid volume. The fluid volume is limited when the fluid chamber is at capacity and when the fluid chamber is empty. The hard stops are modeled with finite stiffness and damping. This means that it is possible for the fluid volume to become

negative or greater than the fluid chamber capacity, depending on the values of the hard-stop stiffness coefficient and the accumulator inlet pressure.



The diagram represents a spring-loaded accumulator. The fluid chamber is on the left and the spring is on the right. The distance between the left side and the spring defines the fluid volume (V_F).

The hard stop contact pressure is modeled with a stiffness term and a damping term. The accumulator spring is assumed to have a linear relationship between the spring pressure and the fluid volume, with pressure balanced at the end of the spring:

$$p_{spr} - p_{pr} = K_{spr} V_F$$

$$p_F = p_{spr} + p_{HS}$$

$$K_{spr} = \frac{p_{\max} - p_{pr}}{V_C}$$

$$p_{HS} = \begin{cases} K_S (V_F - V_C) + K_d q_F^+ (V_F - V_C) & \text{if } V_F \geq V_C \\ K_S V_F - K_d q_F^- V_F & \text{if } V_F \leq 0 \\ 0 & \text{otherwise} \end{cases}$$

$$q_F^+ = \begin{cases} q_F & \text{if } q_F \geq 0 \\ 0 & \text{otherwise} \end{cases}$$

$$\bar{q_F} = \begin{cases} q_F & \text{if } q_F \le 0 \\ 0 & \text{otherwise} \end{cases}$$

where

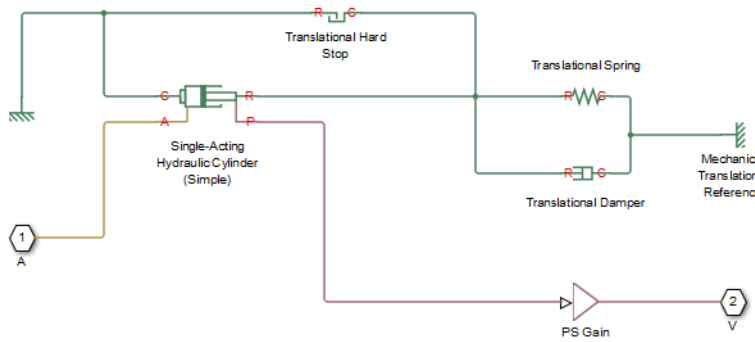
V_F	Volume of fluid in the accumulator
V_{init}	Initial volume of fluid in the accumulator
V_C	Fluid chamber capacity
p_F	Pressure at the accumulator inlet (gauge)
p_{pr}	Preload pressure (gauge)
K_{spr}	Spring gain coefficient
p_{max}	Pressure needed to fully fill the accumulator
p_{spr}	Pressure developed by the spring
p_{HS}	Hard-stop contact pressure
K_s	Hard-stop stiffness coefficient
K_d	Hard-stop damping coefficient
q_F	Fluid flow rate into the accumulator, which is positive if fluid flows into the accumulator

The flow rate into the accumulator is the rate of change of the fluid volume:

$$q_F = \frac{dV_F}{dt}$$

At $t = 0$, the initial condition is $V_F = V_{init}$, where V_{init} is the value you assign to the **Initial fluid volume** parameter.

The Spring-Loaded Accumulator block does not consider loading on the separator. To model additional effects, such as the separator inertia and friction, you can construct a spring-loaded accumulator as a subsystem or a composite component, similar to the block diagram below.



Basic Assumptions and Limitations

- The accumulator spring is assumed to be behave linearly.
- Loading on the separator, such as inertia or friction, is not considered.
- Inlet hydraulic resistance is not considered.
- Fluid compressibility is not considered.

Parameters

- “Parameters Tab” on page 1-1174
- “Variables Tab” on page 1-1175

Parameters Tab

Fluid chamber capacity

Amount of fluid that the accumulator can hold. The default value is `8e-3` m^3.

Preload pressure (gauge)

Spring pressure (gauge) when the fluid chamber is empty. The default value is `10e5` Pa.

Pressure at full capacity (gauge)

Spring pressure (gauge) when the fluid chamber is at capacity. The default value is `30e5` Pa.

Hard-stop stiffness coefficient

Proportionality constant of the hard-stop contact pressure with respect to the fluid volume penetrated into the hard stop. The hard stops are used to restrict the fluid volume between zero and fluid chamber capacity. The default value is `1e10` Pa/m^3.

Hard-stop damping coefficient

Proportionality constant of the hard-stop contact pressure with respect to the flow rate and the fluid volume penetrated into the hard stop. The hard stops are used to restrict the fluid volume between zero and fluid chamber capacity. The default value is `1e10` Pa*s/m^6.

Variables Tab

Accumulator flow rate

Volumetric flow rate through the accumulator port at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Volume of fluid

Volume of fluid in the accumulator at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Pressure of liquid volume

Gauge pressure in the accumulator at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

Ports

The block has one hydraulic conserving port associated with the accumulator inlet.

The flow rate is positive if fluid flows into the accumulator.

See Also

Gas-Charged Accumulator

Introduced in R2006a

Sudden Area Change

Sudden enlargement or contraction

Library

Local Hydraulic Resistances

Description

The Sudden Area Change block represents a local hydraulic resistance, such as a sudden cross-sectional area change. The resistance represents a sudden enlargement if fluid flows from inlet to outlet, or a sudden contraction if fluid flows from outlet to inlet. The block is based on the Local Resistance block. It determines the pressure loss coefficient and passes its value to the underlying Local Resistance block. The block offers two methods of parameterization: by applying semi-empirical formulas (with a constant value of the pressure loss coefficient) or by table lookup for the pressure loss coefficient based on the Reynolds number.

If you choose to apply the semi-empirical formulas, you provide geometric parameters of the resistance, and the pressure loss coefficient is determined automatically according to the following equations (see [1] on page 1-1182):

$$K_{SE} = K_{cor}\left(1 - \frac{A_S}{A_L}\right)^2$$

$$K_{SC} = K_{cor} \cdot 0.5\left(1 - \frac{A_S}{A_L}\right)^{0.75}$$

where

K_{SE}	Pressure loss coefficient for the sudden enlargement, which takes place if fluid flows from inlet to outlet
K_{SC}	Pressure loss coefficient for the sudden contraction, which takes place if fluid flows from outlet to inlet
K_{cor}	Correction factor
A_S	Small area
A_L	Large area

If you choose to specify the pressure loss coefficient by a table, you have to provide a tabulated relationship between the loss coefficient and the Reynolds number. In this case, the loss coefficient is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

The pressure loss coefficient, determined by either of the two methods, is then passed to the underlying Local Resistance block, which computes the pressure loss according to the formulas explained in the reference documentation for that block. The flow regime is checked in the underlying Local Resistance block by comparing the Reynolds number to the specified critical Reynolds number value, and depending on the result, the appropriate formula for pressure loss computation is used.

The Sudden Area Change block is bidirectional and computes pressure loss for both the direct flow (sudden enlargement) and return flow (sudden contraction). If the loss coefficient is specified by a table, the table must cover both the positive and the negative flow regions.

Connections A and B are conserving hydraulic ports associated with the block inlet and outlet, respectively.

The block positive direction is from port A to port B. This means that the flow rate is positive if fluid flows from A to B, and the pressure loss is determined as $p = p_A - p_B$.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- If you select parameterization by the table-specified relationship $K=f(Re)$, the flow is assumed to be turbulent.

Parameters

Model parameterization

Select one of the following methods for block parameterization:

- `By semi-empirical formulas` — Provide geometrical parameters of the resistance. This is the default method.
- `Tabulated data – Loss coefficient vs. Reynolds number` — Provide tabulated relationship between the loss coefficient and the Reynolds number. The loss coefficient is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods. The table must cover both the positive and the negative flow regions.

Small diameter

Internal diameter of the small port, **A**. The default value is `0.01` m.

Large diameter

Internal diameter of the large port, **B**. The default value is `0.02` m. This parameter is used if **Model parameterization** is set to `By semi-empirical formulas`.

Correction coefficient

Correction factor used in the formula for computation of the loss coefficient. The default value is `1`. This parameter is used if **Model parameterization** is set to `By semi-empirical formulas`.

Laminar transition specification

If **Model parameterization** is set to `By semi-empirical formulas`, select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `110`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Reynolds number vector

Specify the vector of input values for Reynolds numbers as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values are `[-4000, -3000, -2000, -1000, -500, -200, -100, -50, -40, -30, -20, -15, -10, 10, 20, 30, 40, 50, 100, 200, 500, 1000, 2000, 4000, 5000, 10000]`. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Loss coefficient vector

Specify the vector of the loss coefficient values as a one-dimensional array. The loss coefficient vector must be of the same size as the Reynolds numbers vector. The default values are `[0.25, 0.3, 0.65, 0.9, 0.65, 0.75, 0.90, 1.15, 1.35, 1.65, 2.3, 2.8, 3.10, 5, 2.7, 1.8, 1.46, 1.3, 0.9, 0.65, 0.42, 0.3, 0.20, 0.40, 0.42, 0.25]`. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) block reference page. This parameter is used if **Model parameterization** is set to `By loss coefficient vs. Re table`.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Interpolation method**
- **Extrapolation method**
- **Laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the resistance inlet.

B

Hydraulic conserving port associated with the resistance outlet.

References

[1] Idelchik, I.E., *Handbook of Hydraulic Resistance*, CRC Begell House, 1994

See Also

Elbow | Gradual Area Change | Local Resistance | Pipe Bend | T-junction

Introduced in R2006b

Sudden Area Change (TL)

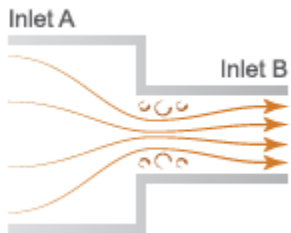
Sudden expansion or contraction in flow area

Library

Thermal Liquid/Pipes & Fittings

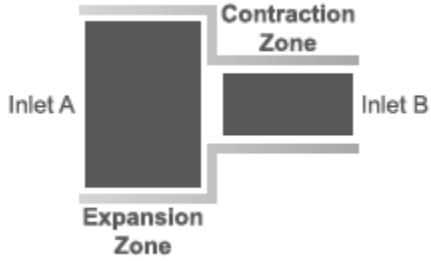
Description

The Sudden Area Change (TL) block models the minor pressure losses due to a sudden change in flow cross-sectional area. The area change is a contraction from port A to port B and an expansion from port B to port A. This component is adiabatic. It does not exchange heat with its surroundings.



Sudden Area Change Schematic

The pressure drop across a sudden expansion is primarily due to turbulence mixing in the expansion zone. Across a sudden contraction, it is primarily due to flow detachment at the contraction zone entrance. The figure shows the expansion and contraction zones of the sudden area change.



Mass Balance

The mass conservation equation in the sudden area change is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- $\dot{m}_A$ and $\dot{m}_B$ are the mass flow rates into the sudden area change through ports A and B.

Momentum Balance

The momentum conservation equation in the sudden area change is

$$p_A - p_B = \frac{\dot{m}^2}{2\rho}\left(\frac{1}{S_B^2} - \frac{1}{S_A^2}\right) + \phi_{Loss},$$

where:

- p_A and p_B are the pressures at ports A and B.
- $\dot{m}$ is the average mass flow rate.
- ρ is the average fluid density.
- S_A and S_B are the flow cross-sectional areas at ports A and B.
- Φ_{Loss} is the mechanical energy loss due to the sudden area change.

The mechanical energy loss is

$$\phi_{Loss} = K_{Loss} \frac{\dot{m}^2}{2\rho S_B^2},$$

where:

- K_{Loss} is the loss coefficient.

If the **Loss coefficient specification** parameter is set to `Semi-empirical formulation`, the loss coefficient for a sudden expansion is computed as

$$K_{Loss} = K_e \left(1 - \frac{S_B}{S_A}\right)^2,$$

while for a sudden contraction it is computed as

$$K_{Loss} = \frac{K_c}{2}\left(1 - \frac{S_B}{S_A}\right),$$

where:

- K_{e} is the correction factor in the expansion zone.
- K_{c} is the correction factor in the contraction zone.

In the transition zone between sudden expansion and sudden contraction behavior, the loss coefficient is smoothed through a cubic polynomial function:

$$K_{Loss} = K_e \left(1 - \frac{S_B}{S_A}\right)^2 + \lambda \left[\frac{K_c}{2}\left(1 - \frac{S_B}{S_A}\right) - K_e \left(1 - \frac{S_B}{S_A}\right)^2\right],$$

where

$$\lambda = 3\bar{\dot{m}}^2 - 2\bar{\dot{m}}^3,$$

and

$$\dot{m}_{Cr} = \mathrm{Re}_{Cr} \sqrt{\frac{\pi}{4} S_B \mu}.$$

If the **Loss coefficient specification** parameter is set to `Tabulated data — Loss coefficient vs. Reynolds number`, the block obtains the loss coefficient from tabular data provided as a function of the Reynolds number.

Energy Balance

The energy conservation equation in the sudden area change is

$$\phi_A + \phi_B = 0,$$

where:

- Φ_A and Φ_B are the energy flow rates into the sudden area change through ports A and B.

Assumptions and Limitations

- The flow is incompressible. The fluid density is assumed constant in the sudden area change.

Parameters

Geometry Tab

Cross-sectional area at port A

Area normal to the direction of flow at inlet A. This value must be greater than the cross-sectional area at B. The default value is `2e-2` m^2.

Cross-sectional area at port B

Area normal to the direction of flow at inlet B. This value must be smaller than the cross-sectional area at A. The default value is `1e-2`.

Characteristic longitudinal length

Average distance traversed by the fluid from inlet A to inlet B. This value must be greater than zero. The default value is `0.1` m.

Parameterization Tab

Loss coefficient specification

Parameterization for calculating the loss coefficient due to the sudden area change. Select `Semi-empirical formulation` to automatically compute the loss coefficient from the cross-sectional areas at ports A and B. Select `Tabulated data — Loss coefficient vs. Reynolds number` to specify a 1-D lookup table for the loss coefficient with respect to the flow Reynolds number. The default setting is `Tabulated data — Loss coefficient vs. Reynolds number`.

Contraction correction factor

Scaling factor for adjusting the loss coefficient value in the contraction portion of the sudden area change. The block multiplies the loss coefficient factor calculated from the semi-empirical expression by this factor. This parameter is visible only when the **Loss coefficient specification** parameter is set to `Semi-empirical formulation`. The default value is `1`.

Expansion correction factor

Scaling factor for adjusting the loss coefficient value in the expansion portion of the sudden area change. The block multiplies the loss coefficient factor calculated from the semi-empirical expression by this factor. This parameter is visible only when the **Loss coefficient specification** parameter is set to `Semi-empirical formulation`. The default value is `1`.

Critical Reynolds number

Reynolds number at which flow transitions between laminar and turbulent regimes in the contraction portion of the sudden area change. This parameter is visible only when the **Loss coefficient specification** parameter is set to `Semi-empirical formulation`. The default value is `10`.

Reynolds number vector

Vector of Reynolds numbers with which to build a loss coefficient lookup table. You specify the **Contraction loss coefficient vector** and **Expansion loss coefficient vector** parameters at these Reynolds numbers.

This parameter is visible only when the **Loss coefficient specification** parameter is set to `Tabulated data — Loss coefficient vs. Reynolds number`. The default vector is a 10-element array ranging from `10.0` to `2000.0`.

Contraction loss coefficient vector

Vector of loss coefficients for the contraction portion of the area change. Specify the loss coefficients at the Reynolds numbers in the **Reynolds number vector** parameter. The block uses the Reynolds number and loss coefficient vectors to construct a 1-D lookup table.

This parameter is visible only when the **Loss coefficient specification** parameter is set to `Tabulated data — Loss coefficient vs. Reynolds number`. The default vector is a 10-element array ranging from `4.0` to `0.2`.

Expansion loss coefficient vector

Vector of loss coefficients for the expansion portion of the area change. Specify the loss coefficients at the Reynolds numbers in the **Reynolds number vector** parameter. The block uses the Reynolds number and loss coefficient vectors to construct a 1-D lookup table.

This parameter is visible only when the **Loss coefficient specification** parameter is set to `Tabulated data — Loss coefficient vs. Reynolds number`. The default vector is a 10-element array ranging from `4.0` to `0.65`.

Variables Tab

Mass flow rate into port A

Mass flow rate into the component through port **A** at the start of simulation. The default value is `1 kg/s`.

Ports

- A — Thermal liquid port representing inlet A
- B — Thermal liquid port representing inlet B

See Also

Pipe (TL)

Introduced in R2016a

Swash Plate

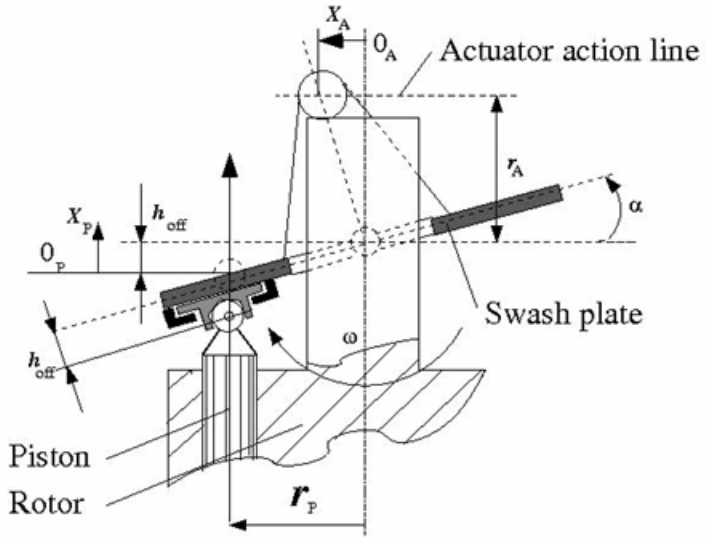
Swash plate mechanism

Library

Pumps and Motors

Description

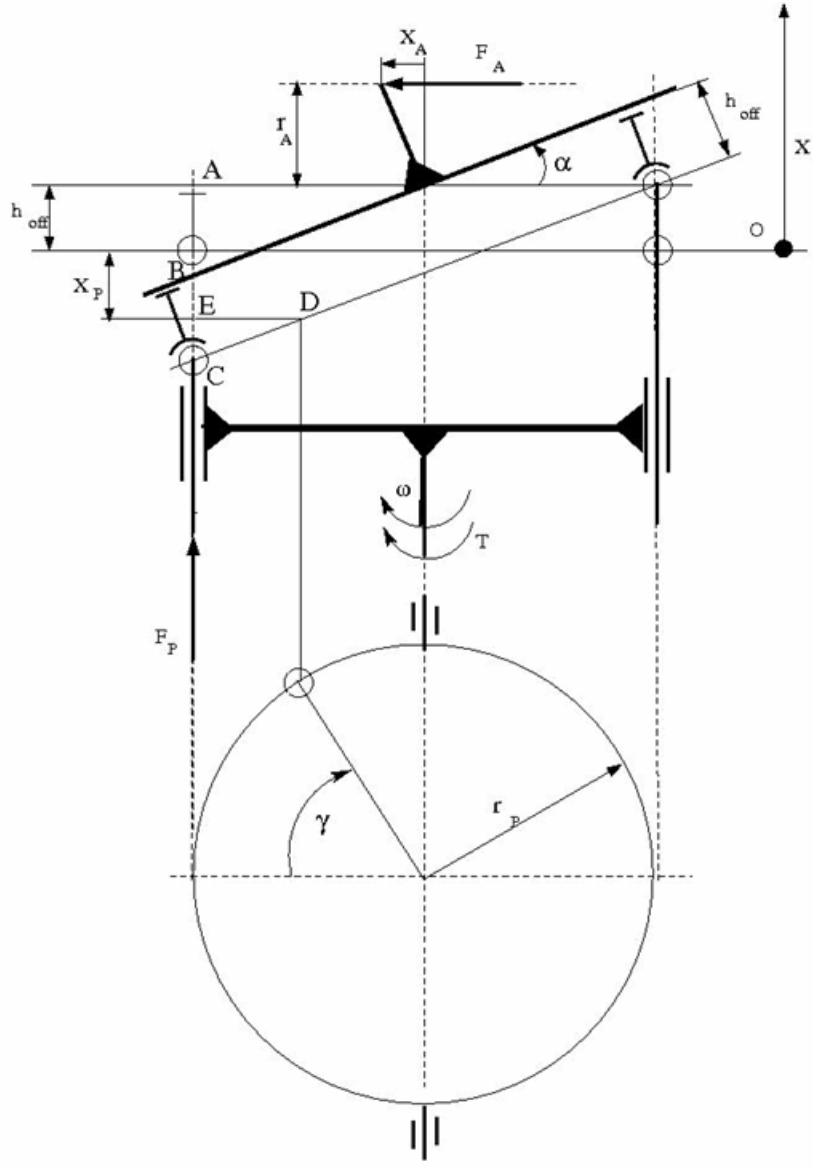
The Swash Plate block is a model of a swash plate mechanism used in axial-piston hydraulic pumps and motors to convert translational motion of a piston into rotational motion of a drive shaft. The mechanism is a combination of a cylinder block or rotor, with the piston fitted inside, and an angled swash plate.



The angular position of the plate is controlled by an actuator, usually a servo cylinder, acting along the actuator action line (marked in the preceding diagram). The distance between the actuator line and the swash plate at zero angle is equal to the actuator arm length r_A. The pistons are offset by h_{off} from the swash plate. The swash plate angle can be changed to increase or decrease the piston stroke. As the rotor turns, the piston slides on the swash plate, which causes the piston to reciprocate. The sliding contact between the piston and the swash plate is maintained by the guideway shown in the diagram.

In a hydraulic machine, the piston moves along the slots in the porting plate (not shown in the preceding diagram) thus being periodically connected to either the intake or the discharge port of the machine. The model simulates a single piston interacting with the swash plate. Mechanically, the mechanism performs summation of two motions, caused by rotation of the swash plate with respect to its axis and rotation of the rotor, and converts them into displacement of the piston.

The following illustration shows the calculation diagram of the mechanism.


X A
F A
r A
h off
X
A
α
h off
O
B
X P
E
D
C
ω
T
F P
γ
r P

The positive direction of the actuator is assumed to be directed to the left from the vertical axis, while the positive direction of the piston motion is directed upward, as it is shown in the diagram. The reference point for both motions corresponds to the plate being perpendicular to the rotor axis. As far as the angular motion of the rotor is concerned, the clockwise rotation (looking from the top) is assumed to be positive.

The piston displacement at arbitrary rotor rotation angle γ and actuator displacement x_A is determined from the relationship:

$$x_P = -\left(AB - h_{off} + BC + DE \cdot tg\alpha\right)$$

$$tg\alpha = \frac{x_A}{r_A}$$

which, after minor rearrangement, yields:

$$x_P = -\frac{r_P}{r_A} x_A \cos\left(\gamma + \beta\right) - h_{off}\left(\sqrt{1 + \frac{x_A^2}{r_A^2}} - 1\right) \tag{1-11}$$

$$\gamma = \omega \cdot t$$

where

x_P	Piston displacement
x_A	Actuator displacement
r_P	Cylinder block pitch radius
r_A	Actuator arm
γ	Rotor angle of rotation
h_{off}	Piston offset
β	Piston phase angle
ω	Rotor angular velocity
t	Time

The kinematic relationship between piston and actuator displacement must be converted into a relationship between the basic Across variables for the physical network representation, that is, velocities:

$$V_P = \frac{dx_P}{dt} = -\frac{r_P}{r_A} V_A \cos(\gamma+\beta) + \frac{r_P}{r_A} x_A \omega \sin(\gamma+\beta) - \frac{h_{off}}{r_A^2} \frac{x_A V_A}{\sqrt{1+\frac{x_A^2}{r_A^2}}}$$

where

V_P	Piston velocity
V_A	Actuator velocity

The inertia properties of the mechanism are assumed to be negligible. Therefore, any state of the mechanism can be considered as equilibrium. According to the principle of virtual work, if the system is in equilibrium the aggregate work of all the applied forces and torques on the system virtual displacements is equal to zero. For the mechanism under consideration, the principle of virtual work yields:

$$F_P \delta x_P + F_A \delta x_A + (T - T_{res}) \delta\gamma = 0 \tag{1-12}$$

where

F_P	Piston force
F_A	Actuator force
T	Rotor torque
T_{res}	Resistance torque acting on the rotor due to viscous friction

Virtual displacements δx_P, δx_A, and $\delta\gamma$ are determined from “Equation 1-11” and substituted into “Equation 1-12”. Since virtual displacements δx_A and $\delta\gamma$ are independent, any of them can be equal to zero. As a result, we obtain two more equations of the mechanism:

$$F_A = \frac{r_P}{r_A} F_P \cos(\gamma+\beta) + \frac{h_{off}}{r_A^2} \frac{x_A}{\sqrt{1+\frac{x_A^2}{r_A^2}}} F_P$$

$$T = -\frac{r_P}{r_A} x_A \cdot F_P \cdot \sin(\gamma+\beta) + T_{res}$$

The resistance torque is:

$$T_{res} = \mu \cdot r_P \cdot \omega$$

where μ is the viscous friction coefficient in the contact between the piston and the plate.

The final set of equations describing the swash plate mechanism takes the form:

$$V_P = \frac{dx_P}{dt} = -\frac{r_P}{r_A} V_A \cos(\gamma + \beta) + \frac{r_P}{r_A} x_A \omega \sin(\gamma + \beta) - \frac{h_{off}}{r_A^2} \frac{x_A V_A}{\sqrt{1 + \frac{x_A^2}{r_A^2}}}$$

$$F_A = \frac{r_P}{r_A} F_P \cos(\gamma + \beta) + \frac{h_{off}}{r_A^2} \frac{x_A}{\sqrt{1 + \frac{x_A^2}{r_A^2}}} F_P$$

$$T = -\frac{r_P}{r_A} x_A \cdot F_P \cdot \sin(\gamma + \beta) + \mu \cdot r_P \cdot F_P$$

Connection P is a mechanical translational conserving port associated with the tip of the piston. Connection S is a mechanical rotational conserving port associated with the drive shaft, which is connected to the cylinder block. Connection A is a mechanical translational conserving port associated with the swash plate actuator.

Basic Assumptions and Limitations

- The model accounts for the viscous friction in the piston-plate contact.
- No inertial effects are considered.
- The plate angular displacements are considered to be small.
- The joint between the piston and the plate permanently maintains contact between the piston and the plate.

Parameters

Swash plate actuator arm

The distance between the actuator line of action and the center of the rotor rotation. The parameter must be greater than zero. The default value is `0.065` m.

Piston pitch radius

The radius of the pitch circle where the pistons are located, that is, the cylinder block pitch radius. The parameter must be greater than zero. The default value is `0.05` m.

Piston offset from the plate

The offset between the piston tip and the swash plate. The parameter must be greater than or equal to zero. The default value is `0`.

Phase angle

Parameter sets the piston initial angular position with respect to the reference point which corresponds to the zero angle. The default value is `0`.

Actuator initial displacement

Parameter sets the initial position of the actuator with respect to the reference point which corresponds to the plate being perpendicular to the axis of rotor rotation. The default value is `0`.

Viscous friction coefficient

The parameter specifies the coefficient of viscous friction in the contact between the piston and the plate. The parameter must be greater than zero. The default value is `50` N/(m/s).

Ports

The block has the following ports:

`A`

Mechanical translational conserving port associated with the actuator joint.

`P`

Mechanical translational conserving port associated with the tip of the piston.

`S`

Mechanical rotational conserving port associated with the drive shaft.

Examples

The Hydraulic Axial-Piston Pump with Load-Sensing and Pressure-Limiting Control example models a test rig designed to investigate interaction between an axial-piston pump and a typical control unit, simultaneously performing the load-sensing and pressure-limiting functions. To assure required accuracy, the model of the pump must account for such features as interaction between pistons, swash plate, and porting plate, which makes it necessary to build a detailed pump model.

See Also

Porting Plate Variable Orifice

Introduced in R2011a

T-junction

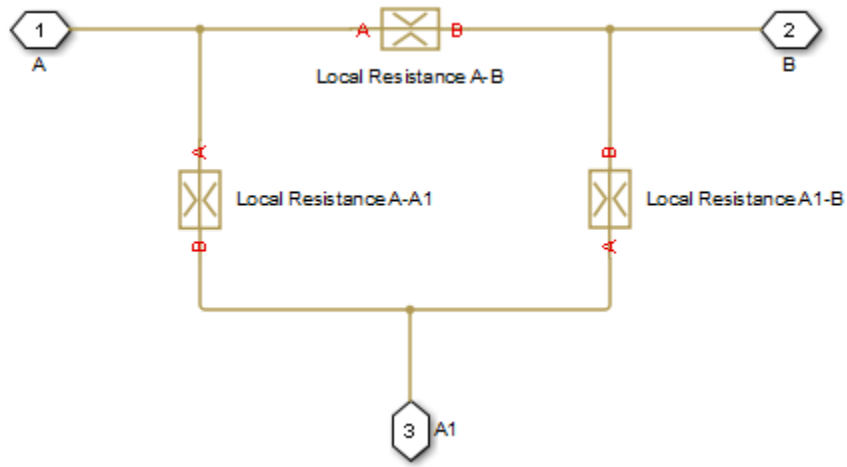
Hydraulic resistance of T-junction in pipe

Library

Local Hydraulic Resistances

Description

The T-junction block represents a T-junction (wye connection) consisting, in general, of a main run and a branch merging to the main run. The junction as a hydraulic resistance is built of three Local Resistance blocks, as shown in the following diagram.



To specify pressure loss for all possible flow directions, you have to provide six pressure loss coefficients. The flow regime is checked in the underlying Local Resistance blocks by

comparing the Reynolds number to the specified critical Reynolds number value, and depending on the result, the appropriate formula for pressure loss computation is used. For more information, see the reference documentation for the Local Resistance block.

The block positive direction is from port A to port B, from port A to port A1, and from port A1 to port B.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.

Parameters

Geometry Tab

Main pipe diameter

The internal pipe diameter of the main run. The default value is `0.01` m.

Branch pipe diameter

The internal pipe diameter of the branch. The default value is `0.01` m.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter

value in hydraulics textbooks. The default value is `120`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Pressure Loss Tab

A-B pressure loss coefficient

The pressure loss coefficient between ports A and B when fluid flows in the direction from A to B. The default value is `1.12`.

B-A pressure loss coefficient

The pressure loss coefficient between ports A and B when fluid flows in the direction from B to A. The default value is `1.12`.

A-A1 pressure loss coefficient

The pressure loss coefficient between ports A and A1 when fluid flows in the direction from A to A1. The default value is `1.36`.

A1-A pressure loss coefficient

The pressure loss coefficient between ports A and A1 when fluid flows in the direction from A1 to A. The default value is `1.65`.

A1-B pressure loss coefficient

The pressure loss coefficient between ports A1 and B when fluid flows in the direction from A1 to B. The default value is `1.6`.

B-A1 pressure loss coefficient

The pressure loss coefficient between ports A1 and B when fluid flows in the direction from B to A1. The default value is `1.8`.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`A`

Hydraulic conserving port associated with the main run inlet.

`B`

Hydraulic conserving port associated with the main run outlet.

`A1`

Hydraulic conserving port associated with the branch inlet.

See Also

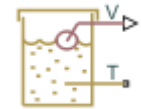
Elbow | Gradual Area Change | Local Resistance | Pipe Bend | Sudden Area Change

Introduced in R2006b

Tank

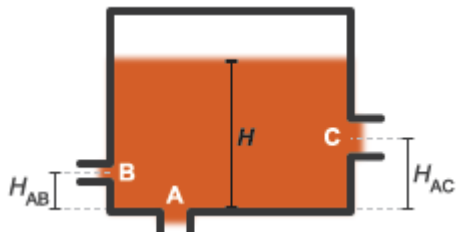
Pressurized fluid container with variable fluid level

Library: Simscape / Fluids / Hydraulics (Isothermal) / Low-Pressure Blocks



Description

The Tank block models a pressurized fluid container with a variable fluid level. The tank has an optional number of ports ranging from one to three, with each number corresponding to a block variant. The tank pressurization is fixed regardless of changes to fluid volume or fluid level. The tank is vented if the pressurization is set to zero—a setting corresponding to an internal pressure equal to atmospheric pressure.



Tank Ports and Port Elevations

The block accounts for differences in port elevation. Hydrostatic pressure is computed separately for each port from the fluid level and port elevation. The hydrostatic pressure adds to the pressurization of the tank—increasing the port pressure as the fluid level rises. The block accounts also for minor pressure losses at the ports due to filters, fittings, and other local flow resistances. To modulate these losses, the block provides a pressure loss coefficient for each port.

Changing the Number of Ports

To change the number of ports in the tank, you must change the active block variant. You can do this from the context-sensitive menu of the block. Right-click the block to open the menu and select **Simscape > Block choices** to view or change the active block variant. Variant options include:

- `One inlet` (default) — Exposes one hydraulic conserving port, **T**.
- `Two inlets` — Exposes hydraulic conserving ports **A** and **B**.
- `Three inlets` — Exposes hydraulic conserving ports **A**, **B**, and **C**.

Port **A** in the `Two inlets` and `Three inlets` variants is a direct replacement of port **T** in the `One inlet` variant.

Port Depth and Fluid Level

Port **T** in the `One inlet` variant—or port **A**, its replacement in the `Two inlets` and `Three inlets` variants—is by default located at the bottom of the tank. The depth of this port relative to the top of the fluid volume is equal to the fluid level—the height of the fluid volume:

$$y_{T/A} = H,$$

where:

- $y_{T/A}$ is the depth of port **T** or **A**.
- H is the fluid level in the tank.

Ports **B** and **C** in the `Two inlets` and `Three inlets` variants are assumed to be located above port **A**. The depths of these ports are each computed as the difference between the fluid level and the port elevation above port **A**:

$$y_B = H - H_{AB},$$

and

$$y_C = H - H_{AC},$$

where:

- y_B and y_C are the depths of ports **B**, and **C**.
- H_{AB} and H_{AC} are the elevations of ports **B** and **C** above port **A**.

The fluid level, H, is computed during simulation from the instantaneous fluid volume. This calculation depends on the **Tank volume parameterization** setting. If the parameterization is set to `Constant cross-sectional area` (the default setting), the fluid level is:

$$H = \frac{V}{S},$$

where:

- V is the instantaneous fluid volume in the tank.
- S is the internal cross-sectional area of the tank, specified in the **Tank cross-section area** parameter.

If the **Tank volume parameterization** is set to `Tabulated data – Volume vs. level`, the fluid level is computed by interpolation or extrapolation of tabulated fluid level data specified as a function of fluid volume:

$$H = f(V)$$

The instantaneous fluid volume varies with the volumetric flow rates through the exposed ports. This volume increases if the sum of all flow rates is positive. A flow rate is positive if it is directed from the port to the interior of the tank, that is, if fluid enters the tank. The time rate of change of the fluid volume is defined as:

$$\frac{dV}{dt} = \sum_i q_i,$$

where q_i is the flow rate into the tank through the port denoted by i—**T/A**, **B**, or **C**.

Flow Rate and Pressure Drop

The volumetric flow rate through a port is a function of the pressure drop from that port to the interior of the tank. The flow rate is positive if the pressure drop is positive—that is, if pressure is higher at the port than in the tank—and negative otherwise:

$$q_i = A_i \sqrt{\frac{2}{K_i \rho}} \frac{\Delta p_i}{\left(\Delta p_i^2 + p_{Cr,i}^2\right)^{1/4}},$$

where:

- A_i is the internal cross-sectional area of the port denoted by i (**T/A**, **B**, or **C**):

  $$A_i = \frac{\pi d_i^2}{4},$$

  with d_i as the internal diameter of the port.
- K_i is the pressure loss coefficient specified for the port.
- ρ is the hydraulic fluid density.
- Δp_i is the pressure drop from the port to the interior of the tank.
- $p_{Cr,i}$ is the critical pressure at which the flow through the port switches between laminar and turbulent.

The critical pressure at a port is computed from the critical Reynolds number, internally set to a value of 15, and from the respective port diameter. The critical pressure is modulated by the pressure loss coefficient specified for the port—with this parameter serving as a gain for the critical pressure:

$$p_{Cr,i} = K_i \frac{\rho}{2} \left(\frac{\mathrm{Re}_{Cr}\, \nu}{d_i} \right),$$

where:

- Re_{Cr} is the critical Reynolds number.
- ν (nu) is the hydraulic fluid dynamic viscosity.
- d_i is the internal diameter of the port denoted by i.

The pressure drop from a port to the interior of the tank is computed from the specified **Pressurization** value, the hydrostatic pressure, and the pressure at the port:

$$\Delta p_i = p_{port,i} - \left(p_{press} + p_{elev,i} \right),$$

where:

- $p_{\text{port,i}}$ is the pressure at the port denoted by i.
- p_{press} is the specified tank **Pressurization** value.
- $p_{\text{elev,i}}$ is the hydrostatic pressure at the port.

The hydrostatic pressure at a port is a function of the port elevation above the bottom of the tank (zero for port **T**/**A**):

$$p_{elev,i} = \rho g y_i ,$$

where

- g is the value of the gravitational acceleration at the mean elevation of the system.
- y_{i} is the elevation of the port denoted by i relative to the bottom of the port.

C/C++ Code Generation

This block supports code generation for real-time simulation tasks. Certain blocks and block settings may be more suitable for simulation on a real-time device. For suggestions on how to improve real-time simulation performance, use the Simulink Performance Advisor (Simulink). Suggestions include ways to reduce model complexity and decrease numerical stiffness.

Select **Analysis** > **Performance Tools** > **Performance Advisor** in the Simulink menu bar to open the Performance Advisor. Set the **Activity** parameter to `Execute real-time application` to view suggestions specific to real-time simulation performance. Expand the **Real-Time** node in the tree view pane to select performance checks specific to Simscape products.

Ports

Output

`V` — Fluid volume, `m^3`

physical signal

Physical signal output port reporting the volume of fluid in the tank.

Conserving

`T` — Tank inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the tank inlet.

Dependencies

This port is active when the block variant is in the default setting of `One inlet`. You can change block variants from the block context-sensitive menu. Right-click the block to open the menu and select **Simscape > Block choices**.

`A` — Tank inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing one of multiple tank inlets.

Dependencies

This port is active when the block variant is set to `Two inlets` or `Three inlets`. You can change block variants from the block context-sensitive menu. Right-click the block to open the menu and select **Simscape > Block choices**.

`B` — Tank inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing one of multiple tank inlets.

Dependencies

This port is active when the block variant is set to `Two inlets` or `Three inlets`. You can change block variants from the block context-sensitive menu. Right-click the block to open the menu and select **Simscape > Block choices**.

`C` — Tank inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing one of multiple tank inlets.

Dependencies

This port is active when the block variant is set to `Three inlets`. You can change block variants from the block context-sensitive menu. Right-click the block to open the menu and select **Simscape > Block choices**.

Parameters

Parameters Tab

`Pressurization` — Gauge pressure of the tank volume
`0` Pa (default) | scalar number in units of pressure

Gauge pressure of the tank volume. The tank pressure is held constant at this value during simulation. The default value, `0` Pa, corresponds to a vented tank—one whose internal pressure is equal to the atmospheric pressure of the system.

`Tank volume parameterization` — Choice of parameterization for the tank volume
`Constant cross-sectional area` (default) | `Tabulated data – volume vs. level`

Choice of parameterization for the tank volume. Select `Constant cross-sectional area` to compute the tank volume from the variable fluid level and a constant cross-sectional area. Select `Tabulated data – volume vs. level` to compute the tank volume by interpolation or extrapolation of tabulated data on the fluid volume at discrete fluid levels.

`Tank cross-section area` — Area of the (constant) tank cross-section
`0.8` m^2 (default) | scalar number in units of area

Area of the tank cross-section, assumed constant over the allowed range of fluid levels. The block uses this parameter to compute the volume of fluid inside the tank.

`Fluid volume vector` — Fluid volumes at which to specify the tank fluid levels
16-element row vector with values ranging from `0` to `0.727` m^3 (default) | *N*-element vector with elements in units of volume

Array of fluid volumes for the 1-D lookup table used to compute the tank fluid level. The array must increase from left to right but the intervals between the array element values

need not be uniform. There must be at least two elements for `Linear` interpolation and three elements for `Smooth` interpolation.

Dependencies

This parameter is active when the **Tank volume parameterization** parameter is set to `Tabulated data – volume vs. level`.

`Fluid level vector` — Fluid levels corresponding to the specified fluid volumes

16-element row vector with values ranging from `0` to `0.3` m (default) | *N*-element vector with elements in units of length

Array of fluid levels corresponding to the specified orifice openings. The number of elements in the array must match the number of elements in the **Opening vector** parameter.

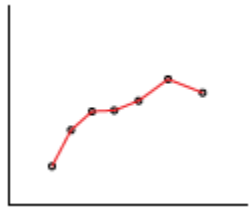
Dependencies

This parameter is active when the **Tank volume parameterization** parameter is set to `Tabulated data – volume vs. level`.

`Interpolation method` — Method of computing values inside the lookup table range

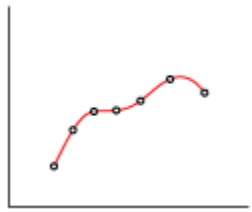
`Linear` (default) | `Smooth`

Method of computing values inside the tabulated data range. The `Linear` method joins adjacent data points with straight line or surface segments with generally discontinuous slope at the segment boundaries. Surface segments are used in the 2-D lookup table specified in the `Pressure-flow characteristic` model parameterization.



The `Smooth` method replaces the straight segments with curved versions that have continuous slope everywhere inside the tabulated data range. The segments form a smooth line or surface passing through all of the tabulated data points without the

discontinuities in first-order derivatives characteristic of the `Linear` interpolation method.



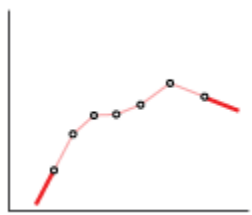
Dependencies

This parameter is active when the **Tank volume parameterization** parameter is set to `Tabulated data — Volume vs. level`.

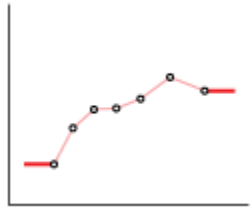
`Extrapolation method` — Method of computing values outside of the lookup table range

`Linear` (default) | `Nearest`

Method of computing values outside of the tabulated data range. The `Linear` method extends the line segment drawn between the last two data points at each end of the data range outward with a constant slope.



The `Nearest` method extends the last data point at each end of the data range outward as a horizontal line with constant value.



Dependencies

This parameter is active when the **Tank volume parameterization** parameter is set to `Tabulated data – Volume vs. level`.

`Inlet pipeline diameter` — Diameter of the tank inlet

`0.02` m (default) | scalar number in units of length

Diameter of the tank inlet. The tank has a single port when this parameter is exposed.

Dependencies

This parameter is active when the block variant is set to `One inlet`.

`Port A/B/C pipeline diameter` — Diameter of tank port A/B/C

`0.02` m (default) | scalar number in units of length

Diameter of tank port **A**/**B**/**C**.

Dependencies

This parameter is active when the block variant is set to `Two inlets` or `Three inlets`.

`Pipeline pressure loss coefficient` — Coefficient used to account for pressure loss at the inlet

`1.2` (default) | unitless scalar number

Empirical coefficient used to account for pressure losses at the tank inlet. This parameter must be greater than zero. The block has a single port when this parameter is exposed.

Dependencies

This parameter is active when the block variant is set to `One inlet`.

`Port A/B/C pressure loss coefficient` — Coefficient used to account for pressure loss at port A/BC
`1.2` (default) | unitless scalar number

Empirical coefficient used to account for pressure losses at port **A**/**B**/**C**. This parameter must be greater than zero.

Dependencies

This parameter is active when the block variant is set to `Two inlets` or `Three inlets`.

`Port B/C height above port A` — Elevation of port B/C above port A
`0.8` m (default) | scalar number in units of length

Elevation of port **B**/**C** above port **A**. The block uses this parameter to account for differences in port elevation.

Dependencies

This parameter is active when the block variant is set to `Two inlets` or `Three inlets`.

`Acceleration due to gravity` — Gravitational acceleration constant
`9.80665` m/s^2 (default) | scalar number in units of acceleration

Gravitational acceleration constant at the mean elevation of the hydraulic system. Changes in gravitational acceleration are assumed negligible over the height of the system.

`Minimum level of fluid` — Lowest fluid level allowed during simulation
`1e-3` m (default) | scalar number in units of length

Lowest fluid level allowed during simulation. The block triggers a simulation warning if the fluid level falls below the specified value.

`Check if fluid level violating minimum valid condition` — Simulation warning mode for minimum valid fluid level
`None` (default) | `Warning`

Simulation warning mode for invalid fluid levels in the tank. Select `Warning` to be notified when the fluid level falls below a minimum specified value.

Variables Tab

`Fluid volume` — Volume of fluid in the tank at time zero
`20` L (default) | scalar number in units of volume

Volume of fluid in the tank at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Fluid level` — Height of fluid volume at time zero
`0.025` m (default) | scalar number in units of length

Height of the fluid volume in the tank at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

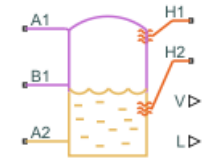
See Also

Constant Head Tank | Reservoir

Tank (G-TL)

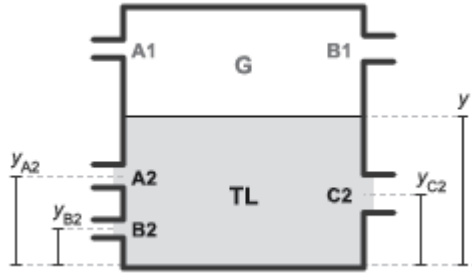
Pressurized tank with variable gas and thermal liquid volumes

Library: Simscape / Fluids / Fluid Network Interfaces / Tanks & Accumulators



Description

The Tank (G-TL) block models the accumulation of mass and energy in a chamber with separate gas and thermal liquid volumes. The total fluid volume is fixed but the individual gas and thermal liquid volumes are free to vary. Two gas ports allow for gas flow and a variable number of thermal liquid ports, ranging from one to three, allow for thermal liquid flow. The thermal liquid ports can be at different elevations.



Tank Inlets and Inlet Heights (*y*)

The tank is pressurized but the pressurization is not fixed. It changes during simulation with the pressure in the gas volume. It rises when the pressure of the gas volume rises and it falls when the pressure of the gas volume falls. The thermal liquid volume is assumed to be at equilibrium with the gas volume and its pressure is therefore the same as that of the gas.

The fluid volumes can exchange energy with other fluid components and with the environment but not with each other. The fluid volumes behave as if they were isolated from each other by an insulated membrane. Energy exchanges with other components occur through gas or thermal liquid ports, while exchanges with the environment occur, strictly in the form of heat, through thermal ports.

Use this block to model components such as drain tanks, in which water condensed from a compressed gas system is trapped at the bottom by gravity and expelled through a drain outlet. Note, however, that neither gas nor thermal liquid domains capture the effects of phase change—and therefore that this block cannot capture the effects of condensation.

Inlet Variants

The number of thermal liquid ports depends on the block variant that is active. To view or change the active variant, right-click the block and select **Simscape > Block Choices**. The `One inlet` variant exposes thermal liquid port **A2**, the `Two inlets` variant adds port **B2**, and the `Three inlets` variant adds port **C2**.

Fluid Volumes

The total volume of the tank is equal to the sum of the gas and thermal liquid volumes that it contains:

$$V_{\mathrm{T}} = V_{\mathrm{L}} + V_{\mathrm{G}},$$

where V is volume and T, L, and G stand for *total*, *liquid*, and *gas*. Because the total volume is fixed, the time rate of change of the gas volume must be the reverse of that measured for the thermal liquid volume:

$$\dot{V}_{\mathrm{G}} = -\dot{V}_{\mathrm{L}}.$$

The time rate of change of the thermal liquid volume is calculated by differentiating the expression:

$$M_{\mathrm{L}} = \rho_{\mathrm{L}} V_{\mathrm{L}},$$

where M is mass and ρ is density. The differentiation gives the mass flow rate into the thermal liquid volume:

$$\dot{M}_{\mathrm{L}} = V_{\mathrm{L}}\dot{\rho}_{\mathrm{L}} + \dot{V}_{\mathrm{L}}\rho_{\mathrm{L}},$$

The time rate of change of the thermal liquid density is:

$$\dot{\rho}_{\mathrm{L}} = \frac{\rho_{\mathrm{L}}}{\beta_{\mathrm{L}}}\dot{p}_{\mathrm{L}} - \alpha_{\mathrm{L}}\rho_{\mathrm{L}}\dot{T}_{\mathrm{L}},$$

where:

- β is the isothermal bulk modulus.
- α is the isobaric thermal expansion coefficient.
- p is the fluid pressure.
- T is the fluid temperature.

Rearranging terms gives the rate of change of the thermal liquid volume and, by extension, of the gas volume:

$$\dot{V}_{\mathrm{G}} = -\dot{V}_{\mathrm{L}} \approx V_{\mathrm{L}}\left(\frac{\dot{p}_{\mathrm{L}}}{\beta_{\mathrm{L}}} - \alpha_{\mathrm{L}}\dot{T}_{\mathrm{L}}\right) - \frac{\dot{M}_{\mathrm{L}}}{\rho_{\mathrm{L}}}$$

Mass Balance

The rate of mass accumulation in each fluid volume is equal to the net mass flow rate into that fluid volume. In the thermal liquid volume:

$$\dot{M}_{\mathrm{L}} = \sum_{i=\mathrm{A2,\ B2,\ C2}} \dot{m}_i,$$

where M_{L} is the rate of mass accumulation in the thermal liquid volume and $\dot{m}_i$ are the individual mass flow rates into that volume through the thermal liquid ports (**A2**, **B2**, and **C2** in the case of the `Three inlets` variant). The rate of mass accumulation contains contributions from pressure, temperature, and volume change:

$$\dot{M}_{\mathrm{L}} = V_{\mathrm{L}}\left(\frac{\rho_{\mathrm{L}}}{\beta_{\mathrm{L}}}\dot{p}_{\mathrm{G}} - \alpha_{\mathrm{L}}\rho_{\mathrm{L}}\dot{T}_{\mathrm{L}}\right) + \dot{V}_{\mathrm{L}}\rho_{\mathrm{L}},$$

where the pressure of the thermal liquid volume is by definition equal to the pressure of the gas volume and the equation is therefore written in terms of the gas pressure. In the gas volume:

$$\dot{M}_{\mathrm{G}} = \sum_{i=\mathrm{A1,\,B1}} \dot{m}_i ,$$

where M_{G} is the rate of mass accumulation in the gas volume and $\dot{m}_i$ are the individual mass flow rates into that volume through the gas ports (**A2** and **B2**). As with the thermal liquid volume, the rate of mass accumulation contains contributions from pressure, temperature, and volume change:

$$\dot{M}_{\mathrm{G}} = \left.\frac{dM}{dp}\right|_{\mathrm{G}} \dot{p}_{\mathrm{G}} + \left.\frac{dM}{dT}\right|_{\mathrm{G}} \dot{T}_{\mathrm{G}} + \dot{V}_{\mathrm{G}} \rho_{\mathrm{G}} ,$$

where the pressure and temperature derivatives depend on the type of gas specified in the Gas Properties (G) block. The derivatives are defined in the equations section of the Translational Mechanical Converter (G) block reference page. Replacing V_{G} with the expression previously obtained for this variable and combining the two expressions for M_{G}:

$$\left.\frac{dM}{dp}\right|_{\mathrm{G}} \dot{p}_{\mathrm{G}} + \left.\frac{dM}{dT}\right|_{\mathrm{G}} \dot{T}_{\mathrm{G}} + \left[V_{\mathrm{L}} \left(\frac{\dot{p}_{\mathrm{G}}}{\beta_{\mathrm{L}}} - \alpha_{\mathrm{L}} \dot{T}_{\mathrm{L}} \right) - \frac{\dot{M}_{\mathrm{L}}}{\rho_{\mathrm{L}}} \right] \rho_{\mathrm{G}} = \sum_{i=\mathrm{A1,\,B1}} \dot{m}_i .$$

Rearranging terms gives the final expression for the mass balance in the gas volume:

$$\dot{p}_{\mathrm{G}} \left(\left.\frac{dM}{dp}\right|_{\mathrm{G}} + \frac{\rho_{\mathrm{G}} V_{\mathrm{L}}}{\beta_{\mathrm{L}}} \right) + \left(\dot{T}_{\mathrm{G}} \left.\frac{dM}{dT}\right|_{\mathrm{G}} - \dot{T}_{\mathrm{L}} \rho_{\mathrm{G}} V_{\mathrm{L}} \alpha_{\mathrm{L}} \right) = \sum_{i=\mathrm{A1,\,B1}} \dot{m}_i + \frac{\rho_{\mathrm{G}}}{\rho_{\mathrm{L}}} \sum_{i=\mathrm{A2,\,B2,\,C2}} \dot{m}_i ,$$

where $\dot{M}_{\mathrm{L}}$ has been replaced with the summation of the mass flow rates into the thermal liquid volume.

Energy Balance

The rate of energy accumulation in each fluid volume is the sum of the energy flow rates through the fluid inlets, the heat flow rate through the corresponding thermal port, and the energy flow rate due to volume changes. For the gas volume:

$$\dot{p}_{\mathrm{G}}\left(\left.\frac{dU}{dp}\right|_{\mathrm{G}}+\rho_{\mathrm{G}}h_{\mathrm{G}}\frac{V_{\mathrm{L}}}{\beta_{\mathrm{L}}}\right)+\left(\dot{T}_{\mathrm{G}}\left.\frac{dU}{dT}\right|_{\mathrm{G}}-\dot{T}_{\mathrm{L}}\rho_{\mathrm{G}}h_{\mathrm{G}}V_{\mathrm{L}}\alpha_{\mathrm{L}}\right)=Q_{\mathrm{H1}}+\sum_{i=\mathrm{A1,\,B1}}\phi_i+\frac{\rho_{\mathrm{G}}}{\rho_{\mathrm{L}}}h_{\mathrm{G}}\sum_{i=\mathrm{A2,\,B2,\,C2}}\dot{m}_i,$$

where:

- U is the total energy of the fluid volume.
- h is the fluid enthalpy.
- Q is the heat flow rate through the thermal port.
- ϕ_{i} are the energy flow rates through the fluid inlets.

As before, the pressure and temperature derivatives depend on the type of gas specified in the Gas Properties (G) block. See the equations section of the Translational Mechanical Converter (G) block reference page for their definitions. For the thermal liquid volume:

$$\dot{p}_{\mathrm{L}}\left.\frac{dU}{dp}\right|_{\mathrm{L}}+\dot{T}_{\mathrm{L}}\left.\frac{dU}{dT}\right|_{\mathrm{L}}=Q_{\mathrm{H2}}+\sum_{i=\mathrm{A2,\,B2,\,C2}}\phi_i+\sum_{i=\mathrm{A2,\,B2,\,C2}}\dot{m}g\left(y(i)-y\right)-h_{\mathrm{L}}\sum_{i=\mathrm{A2,\,B2,\,C2}}\dot{m}_i,$$

where the pressure derivative is:

$$\left.\frac{dU}{dp}\right|_{\mathrm{L}}=-T_{\mathrm{L}}\alpha_{\mathrm{L}}V_{\mathrm{L}},$$

and the temperature derivative is:

$$\left.\frac{dU}{dT}\right|_{\mathrm{L}}=c_{p,\mathrm{L}}\rho_{\mathrm{L}}V_{\mathrm{L}},$$

in which c_{p} is the isobaric specific heat of the thermal liquid inside the tank.

Momentum Balance

Flow resistance due to friction or other causes is ignored in both fluid volumes. The effect of elevation on inlet pressure is also ignored, but only on the gas side. The gas inlet pressures are therefore equal to each other and to the internal pressure of the gas volume:

$$p_{\mathrm{A1}} = p_{\mathrm{B1}} = p_{\mathrm{G}}.$$

The thermal liquid inlet pressures are each a function of inlet depth. The internal pressure of the thermal liquid volume is equal to that of the gas volume ($p_{\mathrm{L}} = p_{\mathrm{G}}$). Including the dynamic pressures ($p_{i,\mathrm{dyn}}$) at the inlets:

$$p_i + p_{i,\mathrm{dyn}} = p_{\mathrm{G}} + \rho_{\mathrm{L}} \left(y - y_i \right) g,$$

where y is the elevation of the thermal liquid surface, y_{i} is the elevation of the thermal liquid inlet, and g is the gravitational constant. The term (y - y_{i}) gives the depth of the thermal liquid inlet with respect to the gas-thermal liquid boundary. The dynamic pressure at each thermal liquid inlet depends on the direction of flow at that inlet:

$$p_{i,\mathrm{dyn}} = \begin{cases} \frac{1}{2} \rho_i v_i^2, & \text{if } \dot{m}_i < 0 \\ 0, & \text{if } \dot{m}_i \geq 0 \end{cases}$$

Ports

Output

V — Thermal liquid volume, m^3

physical signal

Physical signal output port reporting the volume of thermal liquid in the tank.

L — Thermal liquid level, m

physical signal

Physical signal output port reporting the height of the thermal liquid volume relative to the bottom of the tank.

Conserving

A1 — Gas inlet
gas

Opening through which gas can flow into or out of the tank.

B1 — Gas inlet
gas

Opening through which gas can flow into or out of the tank.

A2 — Thermal liquid inlet
thermal liquid

Opening through which thermal liquid can flow into or out of the tank.

B2 — Thermal liquid inlet
thermal liquid

Opening through which thermal liquid can flow into or out of the tank.

Dependencies

This port is active when the block variant is set to `Two inlets` or `Three inlets`. You can change block variants from the block context-sensitive menu. Right-click the block to open the menu and select **Simscape > Block choices**.

C2 — Thermal liquid inlet
thermal liquid

Opening through which thermal liquid can flow into or out of the tank.

Dependencies

This port is active when the block variant is set to `Three inlets`. You can change block variants from the block context-sensitive menu. Right-click the block to open the menu and select **Simscape > Block choices**.

Parameters

Parameters Tab

`Total tank volume` — Combined gas and thermal liquid volume
`10 m^3` (default) | positive scalar with units of volume

Aggregate volume of the gas and thermal liquid portions of the tank.

`Tank volume parameterization` — Choice of parameterization for the thermal liquid volume
`Constant cross-sectional area` (default) | `Tabulated data – volume vs. level`

Choice of parameterization for the thermal liquid volume. Select `Tabulated data – volume vs. level` to compute the thermal liquid volume by interpolation or extrapolation of tabulated data.

`Tank cross-sectional area` — Area of the (constant) tank cross-section
`1 m^2` (default) | scalar number in units of area

Area of the tank cross-section, assumed constant over the allowed range of fluid levels. The block uses this parameter to compute the volume of thermal liquid inside the tank.

`Liquid level vector` — Thermal liquid levels at which to specify the thermal liquid volume
`[0, 3, 5] m` (default) | scalar number in units of length

Vector of thermal liquid levels at which to specify the thermal liquid volume in the tank. The block uses this vector to construct a one-way lookup table for the thermal liquid volume as a function of the thermal liquid level.

Dependencies

This parameter is active when the **Tank volume parameterization** parameter is set to `Tabulated data - volume vs. level`.

`Liquid volume vector` — Thermal liquid volume corresponding to the specified thermal liquid levels
`[0, 4, 6] m^3` (default) | scalar number in units of volume

Vector of thermal liquid volumes corresponding to the values specified in the **Liquid level vector** parameter. The block uses this vector to construct a one-way lookup table for the thermal liquid volume as a function of the thermal liquid level.

Dependencies

This parameter is active when the **Tank volume parameterization** parameter is set to `Tabulated data - volume vs. level`.

`Inlet height at port A2` — Height of the thermal liquid inlet

`0.1 m` (default) | positive scalar with units of length

Height of the thermal liquid inlet.

Dependencies

This parameter is active when the block variant is set to `One inlet`.

`Height vector for inlets A2 and B2` — Heights of the thermal liquid inlets

`[.1, 0] m` (default) | vector with number of elements equal to the number of thermal liquid ports

Vector of heights of the thermal liquid ports relative to the bottom of the tank.

Dependencies

This parameter is active when the block variant is set to `Two inlets`.

`Height vector for inlets A2, B2, and C2` — Heights of the thermal liquid inlets

`[.1, 0.5, 0] m` (default) | vector with number of elements equal to the number of thermal liquid ports

Vector of heights of the thermal liquid ports relative to the bottom of the tank.

Dependencies

This parameter is active when the block variant is set to `Three inlets`.

`Cross-sectional area vector for inlets A1 and B1` — Flow areas of the gas inlets

`[.01, .01] m^2` (default) | vector with two elements

Vector with flow areas of the gas inlets.

`Cross-sectional area at port A2` — Flow area of the thermal liquid inlet

`0.01 m^2` (default) | positive scalar with units of area

Flow area of the thermal liquid inlet.

Dependencies

This parameter is active when the block variant is set to `One inlet`.

`Cross-sectional area vector for inlets A2 and B2` — Flow areas of the thermal liquid inlets

`[0.01, 0.01] m^2` (default) | vector with number of elements equal to the number of thermal liquid ports

Flow areas of the thermal liquid inlets.

Dependencies

This parameter is active when the block variant is set to `Two inlets`.

`Cross-sectional area vector for inlets A2, B2, and C2` — Flow areas of the thermal liquid inlets

`[0.01, 0.01, 0.01] m^2` (default) | vector with number of elements equal to the number of thermal liquid ports

Flow areas of the thermal liquid inlets.

Dependencies

This parameter is active when the block variant is set to `Three inlets`.

`Gravitational acceleration` — Gravitational acceleration constant

`9.81` m/s^2 (default) | scalar number with units of length/time^2

Value of the gravitational acceleration at the elevation of the tank. This constant is assumed constant over the height of the tank.

Variables Tab

`Pressure of gas volume` — Pressure of the gas volume at time zero

`0.101325` MPa (default) | scalar number with units of pressure

Pressure of the gas volume at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with

each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Temperature of gas volume` — Temperature of the gas volume at time zero
`293.15` K (default) | scalar number with units of temperature

Temperature of the gas volume at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Density of gas volume` — Density of the gas volume at time zero
`1.2` kg/m^3 (default) | scalar number with units of mass/volume

Density of the gas volume at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Temperature of liquid volume` — Temperature of the liquid volume at time zero
`293.15` K (default) | scalar number with units of temperature

Temperature of the thermal liquid volume at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Liquid level` — Height of fluid volume at time zero
5 m (default) | scalar number in units of length

Height of the fluid volume in the tank at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Volume of liquid` — Volume of thermal liquid in the tank at time zero
5 m^3 (default) | scalar number in units of volume

Volume of thermal liquid in the tank at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that

conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

`Mass of liquid` — Mass of thermal liquid in the tank at time zero
`5e+3` kg (default) | scalar number in units of mass

Mass of thermal liquid in the tank at time zero. Simscape software uses this parameter to guide the initial configuration of the component and model. Initial variables that conflict with each other or are incompatible with the model may be ignored. Set the **Priority** column to `High` to prioritize this variable over other, low-priority, variables.

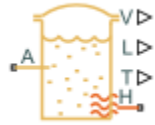
See Also

Tank | Tank (TL)

Introduced in R2017b

Tank (TL)

Thermal liquid container with variable fluid volume



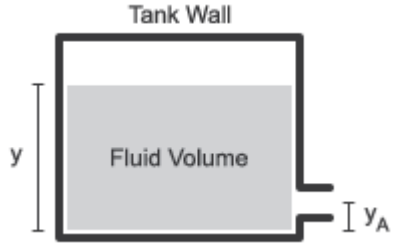
Library

Thermal Liquid/Tanks & Accumulators

Description

The Tank (TL) block models a thermal liquid container with a variable fluid volume. The absolute pressure of the tank fluid volume is assumed constant and equal to the value specified in the block dialog box. In the special case that the tank pressurization is equal to atmospheric pressure, the block represents a vented tank.

The tank can exchange energy with its surroundings, allowing its internal temperature and pressure to evolve over time. Heat transfer occurs via convection, as liquid enters or exits the chamber, and conduction, as thermal energy flows through the tank walls and the liquid itself at the tank inlets.



Tank Schematic

The tank has one default inlet, labeled A, and two optional inlets, labeled B and C. The pressure at the tank inlets is the sum of the constant tank pressurization specified in the block dialog box and the hydrostatic pressure due to the inlet height.

To use the optional tank inlets, right-click the block and select **Simscape > Block choices**. Then, select **Two inlets** to add port B or **Three inlets** to add ports B and C. Port A is always exposed.

The tank model accounts for heat transfer through the tank wall, associated with thermal conserving port H. The temperature defined at this port is the temperature of the tank fluid volume.

Tank Volume

The tank fluid volume is computed from the total fluid mass at each time step:

$$V = \frac{m}{\rho},$$

where:

- V is the tank fluid volume.
- m is the tank fluid mass.
- ρ is the tank fluid density.

Mass Balance

The mass conservation equation in the tank fluid volume is

$$\dot{m} = \dot{m}_A,$$

where:

- $\dot{m}$ is the net mass flow rate into the tank.
- $\dot{m}_A$ is the mass flow rate into the tank fluid volume through the inlet A.

Momentum Balance

The momentum conservation equation in the tank fluid volume is

$$p_A + p_{dyn} = p_{Ref} + \rho g\left(y - y_A\right),$$

where:

- p_A is the fluid pressure at inlet A.
- p_{Ref} is the constant tank pressurization.
- p_{dyn} is the dynamic pressure:

$$p_{dyn} = \begin{cases} 0, & \dot{m}_A \geq 0 \\ \dfrac{\dot{m}_A^2}{2\rho_A S_A^2}, & \dot{m}_A < 0 \end{cases}$$

- ρ_A is the liquid density at port A.
- S_A is the tank inlet area.
- g is the gravitational constant.
- y is the tank level, or height, relative to the tank bottom.
- y_A is the tank inlet elevation relative to the tank bottom.

Energy Balance

The energy conservation equation in the tank fluid volume is

$$m\left(C_p - h\alpha\right)\dot{T} = \phi_A - \dot{m}_A h + Q,$$

where:

- C_p is the fluid thermal capacity.
- α is the fluid isobaric bulk modulus.
- T is the fluid temperature.
- Φ_A is the energy flow rate into the tank through port A.

- h is the fluid enthalpy.
- Q is the thermal energy flow rate into the tank through port H.

Assumptions and Limitations

- The tank pressure is constant and uniform throughout the tank volume. The tank elevation head affects only the inlet pressure calculations.
- Fluid momentum is lost at the tank inlet due to the sudden expansion into the tank volume.

Parameters

Parameters Tab

Pressurization specification

Tank pressurization type. Select `Atmospheric pressure` to model a vented tank. Select `Specified pressure` to model a tank at a custom constant pressure.

Tank pressurization

Absolute pressure in the tank. This parameter is visible only when the **Pressurization specification** parameter is set to `Specified pressure`. The default value, corresponding to atmospheric pressure, is `0.101325` MPa.

Maximum tank capacity

Tank fluid volume in a fully filled state. The default value is `10` m^3.

Tank volume parameterization

Parameterization for calculating the thermal liquid volume as a function of tank level. The default setting is `Constant cross-sectional area`.

Select `Constant cross-sectional area` to set the thermal liquid volume equal to the product of the tank level and cross-sectional area. Select `Tabulated data – Volume vs. level` to directly specify the thermal liquid volume as a function of tank level, e.g., to model a tank of arbitrary geometry.

Tank cross-sectional area

Cross-sectional area for tank fluid volume calculations. This area is assumed uniform along the tank height. This parameter is visible only when the **Pressurization**

specification parameter is set to `Specified pressure`. The default value is `1` m^2.

Liquid level vector

Vector of tank levels at which to specify the thermal liquid volume. The tank level is the thermal liquid height relative to the tank bottom. The block uses this vector to construct a volume-level 1-D lookup table.

This parameter is visible only when the **Pressurization specification** parameter is set to `Tabulated data — Volume vs. level`. The default vector is `[0.0,3.0,5.0]`.

Liquid volume vector

Vector of thermal liquid volumes corresponding to the values specified in the **Liquid level vector** parameter. The block uses this vector to construct a volume-level 1-D lookup table.

This parameter is visible only when the **Pressurization specification** parameter is set to `Tabulated data — Volume vs. level`. The default vector is `[0.0,4.0,6.0]`

Inlet height

Tank inlet elevation for elevation head calculations. If the optional ports are exposed, this parameter is a vector with the inlet elevations. The default value is `0.1` m.

Inlet cross-sectional area

Flow cross-sectional area of the tank inlets. If the optional ports are exposed, this parameter is a vector with the inlet cross-sectional areas. The default value is `0.01` m^2.

Gravitational acceleration

Gravitational acceleration constant for elevation head calculations. The default value is `9.81` m/s^2.

Variables Tab

Liquid level

Height of the thermal liquid volume in the accumulator at the start of simulation. The default value is `5` m.

Volume of liquid

Volume of thermal liquid in the accumulator at the start of simulation. The default value is `5 m^3`.

Mass of liquid

Mass of thermal liquid in the accumulator at the start of simulation. The default value is `5e+3` kg.

Temperature of liquid volume

Temperature in the thermal liquid chamber at the start of simulation. The default value is `293.15` K.

Ports

- A — Thermal liquid conserving port representing the tank inlet A
- B — Thermal liquid conserving port representing the optional tank inlet B
- C — Thermal liquid conserving port representing the optional tank inlet C
- H — Thermal conserving port representing heat transfer through the tank wall
- V — Physical signal output port for the tank fluid volume measurement
- L — Physical signal output port for the tank fluid level measurement
- T — Physical signal output port for the tank fluid temperature measurement

See Also

Gas-Charged Accumulator (TL)

Introduced in R2016a

Temperature Control Valve (G)

Valve with inlet thermostat to regulate flow

Library: Simscape / Fluids / Gas / Valves & Orifices / Flow Control Valves

Description

The Temperature Control Valve (G) block models an orifice with a thermostat as a flow control mechanism. The thermostat contains a temperature sensor and a black-box opening mechanism—one whose geometry and mechanics matter less than its effects. The sensor is at the inlet and it responds with a slight delay, captured by a first-order time lag, to variations in temperature.

When the sensor reads a temperature in excess of a preset activation value, the opening mechanism is actuated. The valve begins to open or close, depending on the chosen operation mode—the first case corresponding to a normally closed valve and the second to a normally open valve. The change in opening area continues up to the limit of the valve's temperature range, beyond which point the opening area is a constant. Within the temperature range, the opening area is a linear function of temperature.

The flow can be laminar or turbulent, and it can reach (up to) sonic speeds. This happens at the vena contracta, a point just past the throat of the valve where the flow is both its narrowest and fastest. The flow then chokes and its velocity saturates, with a drop in downstream pressure no longer sufficing to increase its velocity. Choking occurs when the back-pressure ratio hits a critical value characteristic of the valve. Supersonic flow is not captured by the block.

Control Temperature

The temperature reading at the inlet serves as the control signal for the valve. The greater its rise over the activation temperature, the more the opening area diverges from its normal state—maximally closed in the default **Valve operation** setting of `Opens above activation temperature`, fully open in the alternative setting.

The difference between the sensor temperature reading and the activation temperature is referred to here as the temperature overshoot. For use in the block calculations, this variable is normalized against the temperature regulation range of the valve (that over which the opening area is variable). Its value is computed in this (normalized) form using the expression:

$$\hat{T} = \frac{T_S - T_A}{\Delta T},$$

where T is temperature. The overhead ^ symbol denotes its normalized value while the subscripts `S` and `A` indicate the inlet sensor reading and the activation value (a constant obtained from the block parameter of the same name). The difference ΔT is the temperature regulation of the valve (it too obtained from a block parameter of the same name).

To emulate a real temperature sensor, which can register a shift in temperature only gradually, the block adds a first-order time lag to the temperature reading, T_S. The lag gives the sensor a transient response to variations in temperature. This response is captured in the expression:

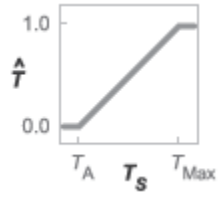
$$\frac{d}{dt}T_S = \frac{T_{In} - T_S}{\tau},$$

where τ is the time required for the sensor to register a step change in temperature. Its value is obtained from the **Sensor time constant** block parameter. The smaller it is, the faster the sensor responds. The subscript `In` denotes the actual inlet temperature at the current time step of the simulation.

The normalized temperature overshoot spans three pressure regions. Below the activation temperature, its value is a constant zero. Above the maximum temperature—the sum of the activation temperature and the temperature regulation range—it is `1`. In between, it varies, as a linear function of the temperature sensor reading, T_S.

The transitions between the regions are sharp and their slopes discontinuous. These pose a challenge to variable-step solvers (the sort commonly used with Simscape models). To precisely capture discontinuities, referred to in some contexts as *zero crossing events*, the solver must reduce its time step, pausing briefly at the time of the crossing in order to recompute its Jacobian matrix (a representation of the dependencies between the state variables of the model and their time derivatives).

This solver strategy is efficient and robust when discontinuities are present. It makes the solver less prone to convergence errors—but it can considerably extend the time needed to finish the simulation run, perhaps excessively so for practical use in real-time simulation. An alternative approach, used here, is to remove the discontinuities altogether.



Normalized temperature overshoot with sharp transitions

To remove the slope discontinuities, the block smoothes them over a small portion of the opening curve. The smoothing, which adds a slight distortion at each transition, ensures that the valve eases into its limiting positions rather than snap (abruptly) into them. The smoothing is optional: you can disable it by setting its time scale to zero. The shape and scale of the smoothing, when applied, derives in part from the cubic polynomials:

$$\lambda_{\mathrm{L}} = 3\bar{T}_{\mathrm{L}}^2 - 2\bar{T}_{\mathrm{L}}^3$$

and

$$\lambda_{\mathrm{R}} = 3\bar{T}_{\mathrm{R}}^2 - 2\bar{T}_{\mathrm{R}}^3,$$

where

$$\bar{T}_{\mathrm{L}} = \frac{\hat{T}}{\Delta T^*}$$

and

$$\bar{T}_{\mathrm{R}} = \frac{\hat{T} - \left(1 - \Delta T^*\right)}{\Delta T^*}.$$

In the equations:

- λ_L is the smoothing expression for the transition from the maximally closed position.
- λ_R is the smoothing expression for the transition from the fully open position.
- Δp^* is the (unitless) characteristic width of the temperature smoothing region:

  $$\Delta T^* = f^* \frac{1}{2},$$

  where f^* is a smoothing factor valued between `0` and `1` and obtained from the block parameter of the same name.

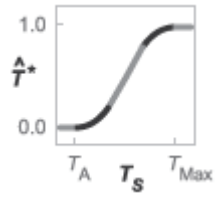
  When the smoothing factor is `0`, the normalized temperature overshoot stays in its original form—no smoothing applied—and its transitions remain abrupt. When it is `1`, the smoothing spans the whole of the temperature regulation range (with the normalized temperature overshoot taking the shape of an *S*-curve).

  At intermediate values, the smoothing is limited to a fraction of that range. A value of `0.5`, for example, will smooth the transitions over a quarter of the temperature regulation range on each side (for a total smooth region of half the regulation range).

The smoothing adds two new regions to the normalized temperature overshoot—one for the smooth transition on the left, another for that on the right, giving a total of five regions. These are expressed in the piecewise function:

$$\hat{T}^* = \begin{cases} 0, & \hat{T} \le 0 \\ \hat{T}\lambda_L, & \hat{T} < \Delta T^* \\ \hat{T}, & \hat{T} \le 1 - \Delta T^* \\ \hat{T}\left(1-\lambda_R\right) + \lambda_R, & \hat{T} < 1 \\ 1 & \hat{T} \ge 1 \end{cases},$$

where the asterisk denotes a smoothed variable (the normalized temperature overshoot). The figure shows the effect of smoothing on the sharpness of the transitions.



Sonic Conductance

As the opening area varies during simulation, so does the mass flow rate through the valve. The relationship between the two variables, however, is indirect. The mass flow rate is defined in terms of the valve's sonic conductance and it is this quantity that the opening area truly determines.

Sonic conductance, if you are unfamiliar with it, describes the ease with which a gas will flow when it is choked—when its velocity is at its theoretical maximum (the local speed of sound). Its measurement and calculation are covered in detail in the ISO 6358 standard (on which this block is based).

Only one value is commonly reported in valve data sheets: one taken at steady state in the fully open position. This is the same specified in the block dialog box (when the **Valve parameterization** setting is `Sonic conductance`). For values across the opening range of the valve, this maximum is scaled by the (normalized) valve opening area:

$$C(S) = \frac{S}{S_{\mathrm{Max}}} C_{\mathrm{Max}},$$

where C is sonic conductance and the subscript `Max` denotes the specified (manufacturer's) value. The sonic conductance varies linearly between C_{Max} in the fully open position and $S_{\mathrm{Leak}} \div S_{\mathrm{Max}} \times C_{\mathrm{Max}}$ in the maximally closed position—a value close to zero and due only to internal leakage between the ports.

Because sonic conductance may not be available (or the most convenient choice for your model), the block provides several equivalent parameterizations. Use the **Valve parameterization** drop-down list to select the best for the data at hand. The parameterizations are:

- `Restriction area`
- `Sonic conductance`
- `Cv coefficient (USCS)`
- `Kv coefficient (SI)`

The parameterizations differ only in the data that they require of you. Their mass flow rate calculations are still based on sonic conductance. If you select a parameterization other than `Sonic conductance`, then the block converts the alternate data—the (computed) opening area or a (specified) flow coefficient—into an equivalent sonic conductance.

The flow coefficients measure what is, at bottom, the same quantity—the flow rate through the valve at some agreed-upon temperature and pressure differential. They differ only in the standard conditions used in their definition and in the physical units used in their expression:

- C_v is measured at a generally accepted temperature of `60 °F` and pressure drop of `1 PSI`; it is expressed in imperial units of `US gpm`. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Cv coefficient (USCS)`.
- K_v is measured at a generally accepted temperature of `15 °C` and pressure drop of `1 bar`; it is expressed in metric units of m^3/h. This is the flow coefficient used in the model when the **Valve parameterization** block parameter is set to `Kv coefficient (SI)`.

If the valve parameterization is set to `Cv Coefficient (USCS)`, the sonic conductance is computed at the maximally closed and fully open valve positions from the **Cv coefficient (SI) at maximum flow** and **Cv coefficient (SI) at leakage flow** block parameters:

$$C = \left(4 \times 10^{-8} C_v\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where C_v is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`. (These are used in the mass flow rate calculations given in the Momentum Balance section.)

If the `Kv coefficient (SI)` parameterization is used instead, the sonic conductance is computed at the same valve positions (maximally closed and fully open) from the **Kv**

coefficient (USCS) at maximum flow and **Kv coefficient (USCS) at leakage flow** block parameters:

$$C = \left(4.758 \times 10^{-8} K_{\mathrm{v}}\right) \mathrm{m}^3/(\mathrm{s\ Pa}),$$

where K_{v} is the flow coefficient value at maximum or leakage flow. The subsonic index, m, is set to `0.5` and the critical pressure ratio, b_{cr}, is set to `0.3`.

For the `Restriction area` parameterization, the sonic conductance is computed (at the same valve positions) from the **Maximum opening area**, and **Leakage area** block parameters:

$$C = \left(0.128 \times 4S/\pi\right) \mathrm{L}/(\mathrm{s\ bar}),$$

where S is the opening area at maximum or leakage flow. The subsonic index, m, is set to `0.5` while the critical pressure ratio, b_{cr} is computed from the expression:

$$0.41 + 0.272\left[\frac{T\left(S_{\mathrm{U}} - S_{\mathrm{L}}\right) + S_{\mathrm{L}}}{S}\right]^{0.25},$$

where S is the valve opening area and the subscripts `U` and `L` denote its values at the upper (`U`) and lower (`L`) bounds of the temperature regulation range. These depend on the setting of the **Valve operation** parameter (`Opens above activation temperature` or `Closes above activation temperature`).

Momentum Balance

The causes of those pressure losses incurred in the passages of the valve are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is assumed to reflect entirely in the sonic conductance of the valve (or in the data of the alternate valve parameterizations).

When the flow is choked, the mass flow rate is a function of the sonic conductance of the valve and of the thermodynamic conditions (pressure and temperature) established at the inlet. The function is linear with respect to pressure:

$$\dot{m}_{\mathrm{ch}} = C\rho_0 p_{\mathrm{in}}\sqrt{\frac{T_0}{T_{\mathrm{in}}}},$$

where:

- C is the sonic conductance inside the valve. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- ρ is the gas density, here at standard conditions (subscript `0`), obtained from the **Reference density** block parameter.
- p is the absolute gas pressure, here corresponding to the inlet (`in`).
- T is the gas temperature at the inlet (`in`) or at standard conditions (`0`), the latter obtained from the **Reference temperature** block parameter.

When the flow is subsonic, and therefore no longer choked, the mass flow rate becomes a nonlinear function of pressure—both that at the inlet as well as the reduced value at the outlet. In the turbulent flow regime (with the outlet pressure contained in the back-pressure ratio of the valve), the mass flow rate expression is:

$$\dot{m}_{\mathrm{tur}} = C\rho_0 p_{\mathrm{in}}\sqrt{\frac{T_0}{T_{\mathrm{in}}}}\left[1-\left(\frac{p_{\mathrm{r}}-b_{\mathrm{cr}}}{1-b_{\mathrm{cr}}}\right)^2\right]^m,$$

where:

- p_{r} is the back-pressure ratio, or that between the outlet pressure (p_{out}) and the inlet pressure (p_{in}):

  $$P_{\mathrm{r}} = \frac{p_{\mathrm{out}}}{p_{\mathrm{in}}}$$

- b_{cr} is the critical pressure ratio at which the flow becomes choked. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).
- m is the *subsonic index*, an empirical coefficient used to more accurately characterize the behavior of subsonic flows. Its value is obtained from the block parameter of the same name or by conversion of other block parameters (the exact source depending on the **Valve parameterization** setting).

When the flow is laminar (and still subsonic), the mass flow rate expression changes to:

$$\dot{m}_{\text{lam}} = C\rho_0 p_{\text{in}} \left[\frac{1-p_{\text{r}}}{1-b_{\text{lam}}}\right] \sqrt{\frac{T_0}{T_{\text{in}}}} \left[1-\left(\frac{b_{\text{lam}}-b_{\text{cr}}}{1-b_{\text{cr}}}\right)^2\right]^m$$

where b_{lam} is the critical pressure ratio at which the flow transitions between laminar and turbulent regimes (obtained from the **Laminar flow pressure ratio** block parameter). Combining the mass flow rate expressions into a single (piecewise) function, gives:

$$\dot{m} = \begin{cases} \dot{m}_{\text{lam}}, & b_{\text{lam}} \le p_{\text{r}} < 1 \\ \dot{m}_{\text{tur}}, & b_{\text{cr}} \le p_{\text{r}} < p_{\text{lam}} \\ \dot{m}_{\text{ch}}, & p_{\text{r}} < b_{\text{Cr}} \end{cases},$$

with the top row corresponding to subsonic and laminar flow, the middle row to subsonic and turbulent flow, and the bottom row to choked (and therefore sonic) flow.

Mass Balance

The volume of fluid inside the valve, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of gas can accumulate there. By the principle of conservation of mass, the mass flow rate into the valve through one port must therefore equal that out of the valve through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the valve through port **A** or **B**. Note that in this block the flow can reach but not exceed sonic speeds.

Energy Balance

The valve is modeled as an adiabatic component. No heat exchange can occur between the gas and the wall that surrounds it. No work is done on or by the gas as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must then always equal zero:

$$\phi_{\text{A}} + \phi_{\text{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the valve through one of the ports (**A** or **B**).

Ports

Conserving

A — Valve entrance

gas

Opening through which the working fluid can enter or exit the valve. The direction of flow depends on the pressure differential established across the valve. Both forward and backward directions are allowed.

B — Valve entrance

gas

Opening through which the working fluid can enter or exit the valve. The direction of flow depends on the pressure differential established across the valve. Both forward and backward directions are allowed.

Parameters

Basic Parameters

`Valve operation` — Sign of the change in opening area induced by warming

`Opens above activation temperature` (default) | `Closes above activation temperature`

Sign of the change in opening area induced by warming. The opening area can expand with a rise in temperature or it can contract. The change begins at an activation temperature and continues with warming conditions throughout the temperature regulation range of the valve.

The default setting corresponds to a normally closed valve that opens with rising temperature; the alternative setting corresponds to a normally open valve that closes with the same.

`Activation temperature` — Temperature at which the opening mechanism is triggered
`330 K` (default) | scalar in units of temperature

Temperature at which the opening mechanism is triggered. Warming above this temperature will either open or close the valve, depending on the setting of the **Valve operation** parameter. The opening area remains variable throughout the temperature regulation range of the valve.

`Temperature regulation range` — Span of the temperature interval over which the opening area is variable
`8 K` (default) | positive scalar in units of temperature

Span of the temperature interval over which the valve opening area varies with temperature. The interval begins at the activation temperature of the valve; it ends at the sum of the same with the regulation range specified here.

`Sensor time constant` — Time for a temperature change to register at the inlet sensor
`1.5 s` (default) | positive scalar in units of time

Characteristic time for a temperature change to register at the inlet sensor. This parameter determines the delay between the onset of a change and a stable measurement of the same (taken as the sensor nears its new steady state). A value of `0` means that the sensor responds instantaneously to a temperature change.

`Valve parameterization` — Method by which to characterize the opening of the valve
`Sonic conductance` (default) | `Cv coefficient (USCS)` | `Kv coefficient (SI)` | `Restriction area`

Choice of ISO method to use in the calculation of mass flow rate. All calculations are based on the **Sonic conductance** parameterization; if a different option is selected, the data specified in converted into equivalent sonic conductance, critical pressure ratio, and subsonic index. See the block description for more information on the conversion.

This parameter determines which measures of valve opening you must specify—and therefore which of those measures appear as parameters in the block dialog box.

`Cross-sectional area at ports A and B` — Area normal to the flow path at the valve ports
`0.01 m^2` (default) | positive scalar in units of area

Area normal to the flow path at the valve ports. The ports are assumed to be the same in size. The flow area specified here should ideally match those of the inlets of adjoining components.

`Laminar flow pressure ratio` — Pressure ratio at which the flow transitions between laminar and turbulent
`0.999` (default) | positive unitless scalar

Pressure ratio at which the flow transitions between laminar and turbulent flow regimes. The pressure ratio is the fraction of the absolute pressure downstream of the valve over that just upstream of it. The flow is laminar when the actual pressure ratio is above the threshold specified here and turbulent when it is below. Typical values range from `0.995` to `0.999`.

`Reference temperature` — Inlet temperature used in the measurement of sonic conductance
`293.15 K` (default) | positive scalar in units of temperature

Absolute temperature used at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

`Reference density` — Inlet density used in the measurement of sonic conductance
`1.185` (default) | positive scalar in units of density

Gas density established at the inlet in the measurement of sonic conductance (as defined in ISO 8778).

Model Parameterization Tab

`Sonic conductance at maximum flow` — Measure of maximum flow rate at reference upstream conditions
`1.6 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the maximum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a maximum when the valve is fully open and the flow velocity is choked (it being saturated at the local speed of sound). This is the value generally reported by manufacturers in technical data sheets.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Sonic conductance at leakage flow` — Measure of minimum flow rate at reference upstream conditions

`1e-5 l/s/bar` (default) | positive scalar in units of volume/time/pressure

Equivalent measure of the minimum flow rate allowed through the valve at some reference inlet conditions, generally those outlined in ISO 8778. The flow is at a minimum when the valve is maximally closed and only a small leakage area—due to sealing imperfections, say, or natural valve tolerances—remains between its ports.

Sonic conductance is defined as the ratio of the mass flow rate through the valve to the product of the pressure and density upstream of the valve inlet. This parameter is often referred to in the literature as the *C-value*.

This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Critical pressure ratio` — Back-pressure ratio, at reference upstream conditions, at which the flow rate is a maximum

`0.3` (default) | positive unitless scalar

Ratio of downstream to upstream absolute pressures at which the flow becomes choked (and its velocity becomes saturated at the local speed of sound). This parameter is often referred to in the literature as the *b-value*. Enter a number greater than or equal to zero and smaller than the **Laminar flow pressure ratio** block parameter.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Subsonic index` — Exponent used to more accurately characterize flow in the subsonic regime

`0.5` (default) | positive unitless scalar

Empirical exponent used to more accurately calculate the mass flow rate through the valve when the flow is subsonic. This parameter is sometimes referred to as the *m-index*. Its value is approximately `0.5` for valves (and other components) whose flow paths are fixed.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Sonic conductance`.

`Cv coefficient (USCS) at maximum flow` — Flow coefficient of the fully open valve expressed in US customary units

`0.4` (default) | positive scalar in units of ft^3/min

Flow coefficient of the fully open valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Cv coefficient (USCS) at leakage flow` — Flow coefficient of the maximally closed valve expressed in US customary units

`1e-6` (default) | positive scalar in units of ft^3/min

Flow coefficient of the maximally closed valve, expressed in the US customary units of ft^3/min (as described in NFPA T3.21.3). This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Cv coefficient (USCS)`.

`Kv coefficient (SI) at maximum flow` — Flow coefficient of the fully open valve expressed in SI units

`0.3` (default) | positive scalar in units of L/min

Flow coefficient of the fully open valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential. This is the value generally reported by manufacturers in technical data sheets.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Kv coefficient (SI) at leakage flow` — Flow coefficient of the maximally closed valve expressed in SI units

`1e-6` (default) | positive scalar in units of L/min

Flow coefficient of the maximally closed valve, expressed in the SI units of `L/min`. This parameter measures the relative ease with which the gas will traverse the valve when driven by a given pressure differential.

The purpose of this parameter is primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Kv coefficient (SI)`.

`Maximum opening area` — Opening area in the fully open position due to sealing imperfections

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the valve in the fully open position, when the valve is at the upper limit of the pressure regulation range. The block uses this parameter to scale the chosen measure

of valve opening—sonic conductance, say, or C_V flow coefficient—throughout the pressure regulation range.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Opening area`.

Leakage area — Opening area in the maximally closed position due to sealing imperfections

`1e-12 m^2` (default) | positive scalar in units of area

Opening area of the valve in the maximally closed position, when only internal leakage between the ports remains. This parameter serves primarily to ensure that closure of the valve does not cause portions of the gas network to become isolated (a condition known to cause problems in simulation). The exact value specified here is less important that its being a (very small) number greater than zero.

Dependencies

This parameter is active and exposed in the block dialog box when the **Valve parameterization** setting is `Opening area`.

Smoothing factor — Amount of smoothing to apply to the valve opening area function

`0` (default)

Amount of smoothing to apply to the opening area function of the valve. This parameter determines the widths of the regions to be smoothed—one located at the fully open position, the other at the fully closed position.

The smoothing superposes on each region of the opening area function a nonlinear segment (a third-order polynomial function, from which the smoothing arises). The greater the value specified here, the greater the smoothing is, and the broader the nonlinear segments become.

At the default value of `0`, no smoothing is applied. The transitions to the maximally closed and fully open positions then introduce discontinuities (associated with zero-crossings), which tend to slow down the rate of simulation.

Variables Tab

Sensor Temperature — Inlet temperature at the start of simulation

`293.15 K` (default) | scalar in units of temperature

Initial condition for the temperature at the inlet of the valve. The block uses a first-order time lag to determine how this temperature will vary during simulation. Set the **Priority** option to **Low** to ignore the value specified here should conflicts with other initial conditions exist.

See Also

Ball Valve (G) | Gate Valve (G) | Variable Orifice ISO 6358 (G)

Introduced in R2018b

Temperature Control Valve (TL)

Flow control valve with temperature-based actuation

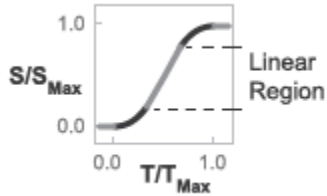
Library

Thermal Liquid/Valves & Orifices/Flow Control Valves

Description

The Temperature Control Valve (TL) block models a valve capable of regulating flow rates based on fluid temperature. The valve begins to open at the activation temperature. It is fully opened at the peak of the temperature regulation range. The block approximates the sensor dynamics with a first-order time lag.

A smoothing function allows the valve opening area to change smoothly between the fully closed and fully open positions. The smoothing function does this by removing the abrupt opening area changes at the zero and maximum ball positions. The figure shows the effect of smoothing on the valve opening area curve.



Opening-Area Curve Smoothing

Valve Opening Area

The valve opening area calculation is based on the linear expression

$$S_{Linear} = \left(\frac{S_{End} - S_{Start}}{T_{Range}} \right) \left(T_{Sensor} - T_{Activation} \right) + S_{Start},$$

where:

- S_{Linear} is the linear valve opening area.
- S_{Start} is the valve opening area at the beginning of the temperature actuation range. This area depends on the **Valve operation** parameter setting:

$$S_{Start} = \begin{cases} S_{Leak}, & \text{Valve opens above activation temperature} \\ S_{Max}, & \text{Valve closes above activation temperature} \end{cases}$$

- S_{End} is the valve opening area at the end of the temperature actuation range. This area depends on the **Valve operation** parameter setting:

$$S_{End} = \begin{cases} S_{Max}, & \text{Valve opens above activation temperature} \\ S_{Leak}, & \text{Valve closes above activation temperature} \end{cases}$$

- S_{Max} is the valve opening area in the fully open position.
- S_{Leak} is the valve opening area in the fully closed position. Only leakage flow remains in this position.
- T_{Range} is the temperature regulation range.
- $T_{\text{Activation}}$ is the minimum temperature required to operate the valve.
- T_{Sensor} is the measured valve temperature.

The valve model accounts for a first-order lag in the measured valve temperature through the differential equation:

$$\frac{d}{dt}\left(T_{Sensor} \right) = \frac{T_{Avg} - T_{Sensor}}{\tau},$$

where:

- T_{Avg} is the arithmetic average of the valve port temperatures,

  $$T_{Avg} = \frac{T_A + T_B}{2},$$

  where T_A and T_B are the temperatures at ports A and B.
- τ is the **Sensor time constant** value specified in the block dialog box.

The valve opening expressions introduce undesirable discontinuities at the fully open and fully closed positions. The block eliminates these discontinuities using polynomial expressions that smooth the transitions to and from the fully open and fully closed positions. The valve smoothing expressions are

$$\lambda_L = 3\overline{T}_L^2 - 2\overline{T}_L^3$$

and

$$\lambda_R = 3\overline{T}_R^2 - 2\overline{T}_R^3$$

where:

$$\overline{T}_L = \frac{T_{Sensor} - T_{Activation}}{\Delta T_{smooth}}$$

and

$$\overline{T}_R = \frac{T_{Sensor} - \left(T_{Activation} + T_{Range} - \Delta T_{smooth}\right)}{\Delta T_{smooth}}.$$

In the equations:

- λ_L is the smoothing expression for the fully closed portion of the valve opening curve.
- λ_R is the smoothing expression applied to the fully open portion of the valve opening curve.
- ΔT_{smooth} is the temperature smoothing region:

$$\Delta T_{smooth} = f_{smooth}\frac{T_{Range}}{2},$$

where f_{smooth} is a smoothing factor between 0 and 1.

The smoothed valve opening area is given by the piecewise conditional expression

$$S_R = \begin{cases} S_{Start}, & T_{Sensor} \le T_{Activation} \\ S_{Start}\left(1-\lambda_L\right) + S_{Linear}\lambda_L, & T_{Sensor} < T_{Activation} + \Delta T_{smooth} \\ S_{Linear}, & T_{Sensor} < T_{Activation} + T_{Range} - \Delta T_{smooth} \\ S_{Linear}\left(1-\lambda_R\right) + S_{End}\lambda_R, & T_{Sensor} < T_{Activation} + T_{Range} \\ S_{End}, & T_{Sensor} \ge T_{Activation} + T_{Range} \end{cases},$$

where:

- S_{R} is the smoothed valve opening area.

Mass Balance

The mass conservation equation in the valve is

$$\dot{m}_A + \dot{m}_B = 0,$$

where:

- $\dot{m}_A$ is the mass flow rate into the valve through port A.
- $\dot{m}_B$ is the mass flow rate into the valve through port B.

Energy Balance

The energy conservation equation in the valve is

$$\phi_A + \phi_B = 0,$$

where:

- ϕ_A is the energy flow rate into the valve through port A.
- ϕ_B is the energy flow rate into the valve through port B.

Momentum Balance

The momentum conservation equation in the valve is

$$p_A - p_B = \frac{\dot{m}\sqrt{\dot{m}^2 + \dot{m}_{cr}^2}}{2\rho_{Avg} C_d^2 S^2}\left[1 - \left(\frac{S_R}{S}\right)^2\right] PR_{Loss},$$

where:

- p_A and p_B are the pressures at port A and port B.
- $\dot{m}$ is the mass flow rate.
- $\dot{m}_{cr}$ is the critical mass flow rate:

  $$\dot{m}_{cr} = \mathrm{Re}_{cr}\, \mu_{Avg} \sqrt{\frac{\pi}{4} S_R}.$$

- ρ_{Avg} is the average liquid density.
- C_d is the discharge coefficient.
- S is the valve inlet area.
- PR_{Loss} is the pressure ratio:

$$PR_{Loss} = \frac{\sqrt{1 - \left(S_R/S\right)^2 \left(1 - C_d^2\right)} - C_d \left(S_R/S\right)}{\sqrt{1 - \left(S_R/S\right)^2 \left(1 - C_d^2\right)} + C_d \left(S_R/S\right)}.$$

Parameters

Parameters Tab

Valve operation

Effect of fluid temperature on valve operation. Options include `Opens above activation temperature` and `Closes above activation temperature`. The default setting is `Opens above activation temperature`.

Activation temperature

Temperature required to actuate the valve. If the **Valve operation** parameter is set to `Opens above activation temperature`, the valve begins to open at the activation temperature. If the **Valve operation** parameter is set to `Closes above activation temperature`, the valve begins to close at the activation temperature. The default value is `330` K.

Temperature regulation range

Temperature change from the activation temperature required to fully open the valve. The default value is `8` K, corresponding to a fully open valve at a temperature of `338` K.

Sensor time constant

Time constant in the first-order equation used to approximate the temperature sensor dynamics. The default value is `1.5` s.

Maximum opening area

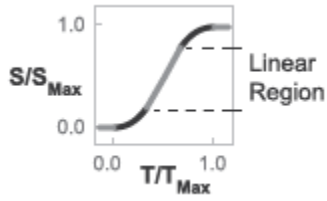
Valve flow area in the fully open position. The default value is `1e-4` m^2.

Leakage area

Area through which fluid can flow in the fully closed valve position. This area accounts for leakage between the valve inlets. The default value is `1e-12` m^2.

Smoothing factor

Portion of the opening-area curve to smooth expressed as a fraction. Smoothing eliminates discontinuities at the minimum and maximum flow valve positions. The smoothing factor must be between `0` and `1`.



Opening-Area Curve Smoothing

A value of `0` corresponds to a linear expression with zero smoothing. A value of `1` corresponds to a nonlinear expression with maximum smoothing. The default value is `0.01`, corresponding to a nonlinear region spanning 1% the size of the full curve..

Cross-sectional area at ports A and B

Area normal to the direction of flow at the valve inlets. This area is assumed the same for all the inlets. The default value is `0.01` m^2.

Characteristic longitudinal length

Approximate length of the valve. This parameter provides a measure of the longitudinal scale of the valve. The default value is `0.1` m^2.

Discharge coefficient

Semi-empirical parameter commonly used as a measure of valve performance. The discharge coefficient is defined as the ratio of the actual mass flow rate through the valve to its theoretical value.

The block uses this parameter to account for the effects of valve geometry on mass flow rates. Textbooks and valve data sheets are common sources of discharge coefficient values. By definition, all values must be greater than 0 and smaller than 1. The default value is `0.7`.

Critical Reynolds number

Reynolds number corresponding to the transition between laminar and turbulent flow regimes. The flow through the valve is assumed laminar below this value and turbulent above it. The appropriate values to use depend on the specific valve geometry. The default value is `12`.

Variables Tab

Sensor temperature

Fluid temperature at the start of simulation. The default value is `293.15` K, corresponding to room temperature.

Mass flow rate into port A

Mass flow rate into the component through port **A** at the start of simulation. The default value is `1 kg/s`.

Ports

- A — Thermal liquid conserving port representing valve inlet A
- B — Thermal liquid conserving port representing valve inlet B

See Also

Ball Valve (TL) | Gate Valve (TL)

Introduced in R2016a

Thermal Liquid Properties (TL)

Preset fluid properties for the simulation of a thermal liquid network
Library: Simscape / Fluids / Thermal Liquid / Utilities

Description

The Thermal Liquid Properties (TL) block applies to a thermal liquid network the properties of a fluid selected from a preset list. Preset fluids include pure water, aqueous mixtures—of saline, glycol and glycerol compounds, commonly used in heat transfer applications as coolants and antifreeze solutions—and fuels such as diesel and aviation-grade Jet A. Use this block as a simple alternative to the Thermal Liquid Settings (TL) block, to define a fluid without having to specify in detail all of its properties. Every thermal liquid network in a model must contain one instance of either of these blocks.

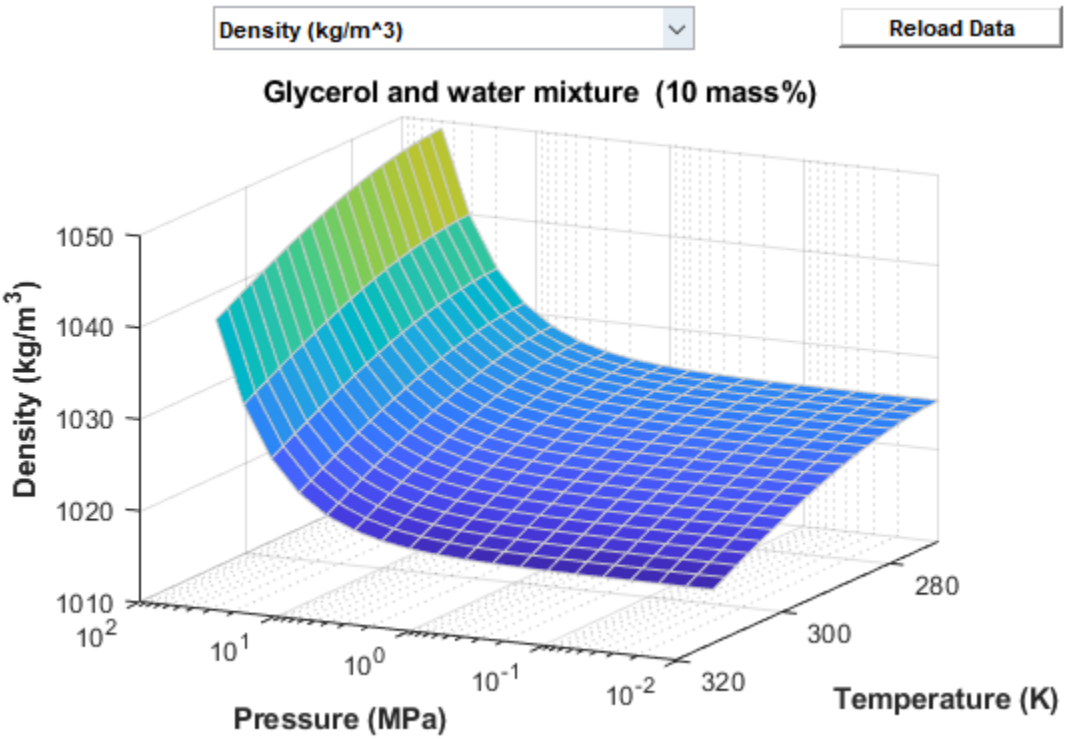
The preset fluid properties are defined in tabular form as functions of temperature and pressure. The table data is sourced from Coolprop—an open-source fluids database—or, in the case of seawater, from a computational model developed by (and proprietary to) MIT. The values of the properties are set during simulation by linear interpolation of the nearest tabulated breakpoints. The effect of concentration is factored into the property calculations for mixtures (with mass or volume fraction providing the necessary measure of concentration).

All the fluid properties commonly specified in the Thermal Liquid Settings (TL) block are defined in the block. These properties include density, the bulk modulus and thermal expansion coefficient, the specific internal energy and specific heat, as well as the kinematic viscosity and thermal conductivity. The properties are valid over a limited region of temperatures and pressures specific to the fluid selected and dependent, in the cases of mixtures, on the concentration specified. Simulation is allowed within this validity region only.

Data Visualization

A data visualization utility provides a means to graph the fluid properties defined in the block. Use it to examine the temperature and pressure dependencies of those properties

or to ascertain the bounds of their validity regions (equal in the visualizations to the bounds of the plots). To open the visualization utility, right-click the block and from the context-sensitive menu select **Fluids > Plot Fluid Properties**. The plot updates automatically upon selection of a fluid property from the drop-down list. Use the **Refresh** button to regenerate the plot whenever the fluid selection or any of its required parameters are changed.



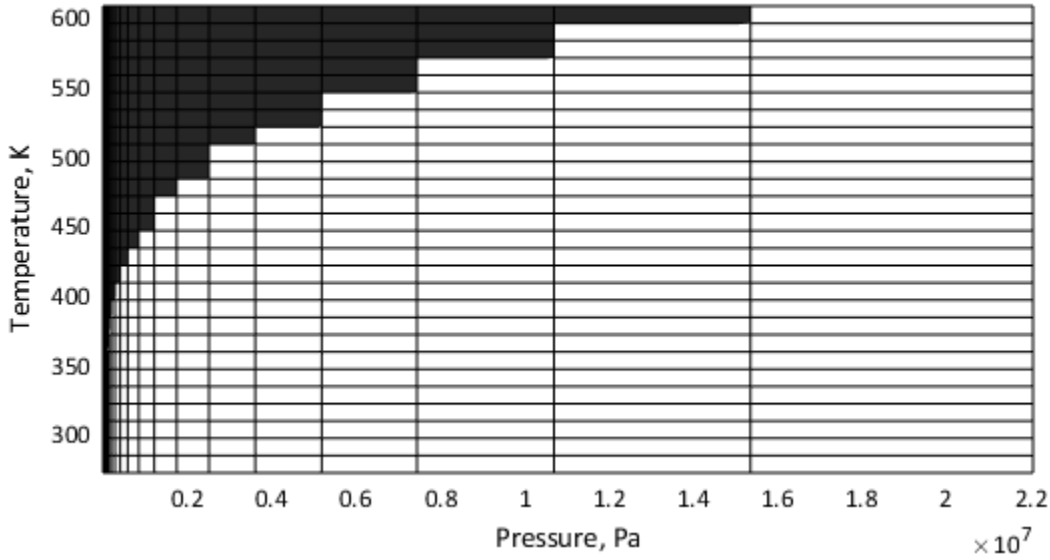
Visualization of density data for a 10% glycerol aqueous mixture

Validity Regions

The validity regions are defined in the block as matrices of zeros and ones. Each row corresponds to a tabulated temperature and each column to a tabulated pressure. A zero denotes an invalid breakpoint and a one a valid breakpoint. These validity matrices are

internal to the block and cannot be modified; they can only be checked (using the data visualization utility of the block).

In most cases, the validity matrices are extracted directly from the tabulated data. Glycol and glycerol mixtures are a partial exception. Their pressure bounds are not available from the Coolprop data (where they are treated as incompressible fluids) and must therefore be obtained explicitly from block parameters. The figure shows an example of a validity region, that of pure water. Shaded squares are outside of the validity region.



The properties of water are valid at temperatures above the triple-point value (`273.160 K`) up to the critical-point value (`647.096 K`). They are valid at pressures above the greater of the triple-point value (`611.657 Pa`) on one hand and the temperature-dependent saturation value on the other, up to the critical-point value (`22,064,000 MPa`). Pressures below the saturation point for a given temperature row are assigned a value of `0` in the validity matrix.

The properties of seawater are valid at temperatures above `0°C` up to `120°C` (`273.15 K` to `393.15 K`); they are valid at pressures above the saturation point up to a maximum value of `12 MPa`. Pressures below the saturation point for a given temperature row (and

at the specified concentration level) are assigned a value of `0` in the validity matrix. Mixture concentrations can range in value from `0` to `0.12` on a mass fraction basis.

The properties of an aqueous ethylene glycol mixture are valid over a temperature domain determined from the mixture concentration; they are valid at pressures within the minimum and maximum bounds specified in the block dialog box (extended horizontally to span the width of the temperature rows).

The lower temperature bound is always the lesser of the minimum temperature extracted from the Coolprop data and the freezing point of the mixture (the mixture must be in the liquid state). The upper temperature bound is always the maximum temperature extracted from the Coolprop data. Mixture concentrations can range in value from `0` to `0.6` if a mass-fraction basis is used, or from `0` to `1` if a volume fraction basis is used.

The properties of an aqueous propylene glycol mixture are valid over the temperature and pressure ranges described for the case of `Ethylene glycol and water mixture`. Mixture concentrations can range in value from `0` to `0.6` if a mass-fraction basis is used, or from `0.1` to `0.6` if a volume fraction basis is used.

The properties of an aqueous glycerol mixture are valid over the temperature and pressure ranges as described for the case of `Ethylene glycol and water mixture`. Mixture concentrations can range in value from `0` to `0.6` on a mass-fraction basis.

The properties of diesel fuel are valid at temperatures above `-34.95°C` up to `417.82°C` (`238.20 K` to `690.97 K`); they are valid at pressures above the saturation point up to a maximum value of `2.29 MPa`. Pressures below the saturation point for a given temperature row are assigned a value of `0` in the validity matrix.

The properties of Jet A fuel are valid at temperatures above `-50.93°C` up to `372.46°C` (`222.22 K` to `645.61 K`); they are valid at pressures above the saturation point up to a maximum value of `2.41 MPa`. Pressures below the saturation point for a given temperature row are assigned a value of `0` in the validity matrix.

Density Calculations

The aqueous mixtures of glycol and glycerol compounds are treated in the Coolprop database as incompressible substances. Their bulk moduli are unavailable from the data

and must instead be obtained from the block parameters (where they are specified as constants). The pressure dependencies of their thermal expansion coefficients are likewise missing and must therefore be calculated (using the bulk modulus provided). Let density be:

$$\rho(T,p) = dT\left(\frac{\partial\rho(T,p)}{\partial T}\right)_p + dp\left(\frac{d\rho(T,p)}{dp}\right)_T,$$

where ρ is density, T is temperature, and p is pressure.

The solution has the form:

$$\rho(T,p) = \rho(T)\exp\left(\frac{p-p_\mathrm{R}}{\beta}\right),$$

where β is the isothermal bulk modulus, and where the subscript R denotes a reference value, here the atmospheric pressure at which the bulk modulus is specified. The partial derivative of density with respect to temperature is:

$$\left(\frac{\partial\rho(T,p)}{\partial T}\right)_p = \left(\frac{\partial\rho(T)}{\partial T}\right)_T \exp\left(\frac{p-p_\mathrm{R}}{\beta}\right).$$

The thermal expansion coefficient is defined as:

$$\alpha(T,p) = -\frac{1}{\rho(T,p)}\left(\frac{\partial\rho(T,p)}{\partial T}\right)_p,$$

Equivalently:

$$\alpha(T,p) = -\frac{1}{\rho(T,p)}\left(\frac{\partial\rho(T)}{\partial T}\right)_T \exp\left(\frac{p-p_\mathrm{R}}{\beta}\right).$$

The block provides the thermal expansion coefficient in this form to the thermal liquid network of which it is a part.

Ports

Conserving

R — Thermal liquid network for which to define the working fluid
thermal liquid

Node identifying the thermal liquid network for which to define the necessary fluid properties. The fluid selected in this block applies to the entire network. No other Thermal Liquid Properties (TL) or Thermal Liquid Settings (TL) block may be connected to the same network.

Parameters

`Thermal liquid fluid list` — Preset fluid whose properties to use in a thermal liquid simulation
`Water` (default) | `Seawater (MIT model)` | `Ethylene glycol and water mixture` | `Propylene glycol and water mixture` | `Glycerol and water mixtureDiesel FuelAviation fuel Jet-A`

Choice of preset fluid whose properties to use in the calculations of the connected thermal liquid network. The fluid properties are automatically set using an internal database derived from the Coolprop fluids library. The fluids provided include pure water, aqueous mixtures, and fuels.

`Dissolved salt mass fraction (salinity)` — Mass of salt divided by the total mass of the saline mixture
`3.5e-3` (default) | positive unitless scalar valued between 0 and 1

Ratio of the mass of salt present in the saline mixture to the total mass of that mixture.

Dependencies

This parameter is active only when `Seawater (MIT model)` is selected as the working fluid.

`Concentration type` — Quantity in terms of which the mixture concentration is defined
`Volume fraction` (default) | `Mass fraction`

Quantity in terms of which to specify the concentration of ethylene glycol in its aqueous mixture. This parameter is active only when either `Ethylene glycol and water mixture` or `Propylene glycol and water mixture` is selected as the working fluid.

`Ethylene glycol volume fraction` — Volume of ethylene glycol divided by that of the aqueous mixture

`0.1` (default) | positive unitless scalar valued between 0 and 1

Volume of ethylene glycol present in the aqueous mixture divided by the total volume of that mixture.

Dependencies

This parameter is active when `Ethylene glycol and water mixture` is selected as the working fluid and `Volume fraction` is selected as the concentration type.

`Ethylene glycol mass fraction` — Mass of ethylene glycol divided by that of the aqueous mixture

`0.1` (default) | positive unitless scalar valued between 0 and 1

Mass of ethylene glycol present in the aqueous mixture divided by the total mass of that mixture.

Dependencies

This parameter is active when `Ethylene glycol and water mixture` is selected as the working fluid and `Mass fraction` is selected as the concentration type.

`Propylene glycol volume fraction` — Mass of propylene glycol divided by that of the aqueous mixture

`0.1` (default) | positive unitless scalar valued between 0 and 1

Volume of propylene glycol present in the aqueous mixture divided by the total volume of that mixture.

Dependencies

This parameter is active when `Propylene glycol and water mixture` is selected as the working fluid and `Volume fraction` is selected as the concentration type.

`Propylene glycol mass fraction` — Mass of propylene glycol divided by that of the aqueous mixture

`0.1` (default) | positive unitless scalar valued between 0 and 1

Mass of propylene glycol present in the aqueous mixture divided by the total mass of that mixture.

Dependencies

This parameter is active when `Propylene glycol and water mixture` is selected as the working fluid and `Mass fraction` is selected as the concentration type.

`Isothermal bulk modulus` — Measure of fluid compressibility specified at constant temperature

`2.1791 GPa` (default) | positive scalar with units of pressure

Bulk modulus of the aqueous mixture at constant temperature. The bulk modulus measures the change in pressure required to produce a fractional change in fluid volume.

Dependencies

This parameter is active when either `Ethylene glycol and water mixture`, `Propylene glycol and water mixture`, or `Glycerol and water mixture` is selected as the working fluid.

`Minimum valid pressure` — Lower bound of the allowed pressure range for the thermal liquid network

`0.01 MPa` (default) | positive scalar with units of pressure

Lower bound of the pressure range allowed in the thermal liquid network connected to this block.

Dependencies

This parameter is active when either `Ethylene glycol and water mixture`, `Propylene glycol and water mixture`, or `Glycerol and water mixture` is selected as the working fluid.

`Maximum valid pressure` — Upper bound of the allowed pressure range for the thermal liquid network network

`50 MPa` (default) | positive scalar with units of pressure

Upper bound of the pressure range allowed in the thermal liquid network connected to this block.

Dependencies

This parameter is active when either `Ethylene glycol and water mixture`, `Propylene glycol and water mixture`, or `Glycerol and water mixture` is selected as the working fluid.

`Atmospheric pressure` — Absolute pressure in the environment of the thermal liquid network network

`0.101325 MPa` (default) | positive scalar with units of pressure

Absolute pressure of the external environment in which the thermal liquid network is assumed to run. The default value is the standard atmospheric pressure measured at sea level on Earth.

References

[1] Massachusetts Institute of Technology (MIT), *Thermophysical properties of seawater database*. http://web.mit.edu/seawater.

[2] K.G. Nayar, M.H. Sharqawy, L.D. Banchik, J.H. Lienhard V, Thermophysical properties of seawater: A review and new correlations that include pressure dependence, *Desalination*, Vol. 390, pp. 1-24, 2016.

[3] M.H. Sharqawy, J.H. Lienhard V, S.M. Zubair, Thermophysical properties of seawater: A review of existing correlations and data, *Desalination and Water Treatment*, Vol. 16, pp. 354-380.

[4] I.H. Bell, J. Wronski, S. Quoilin, V. Lemort, Pure and Pseudo-pure Fluid Thermophysical Property Evaluation and the Open-Source Thermophysical Property Library CoolProp, *Industrial & Engineering Chemistry Research*, Vol. 53 (6), pp. 2498–2508, 2014.

See Also

Thermal Liquid Settings (TL)

Introduced in R2018a

Valve Actuator

Simplified model of valve driver

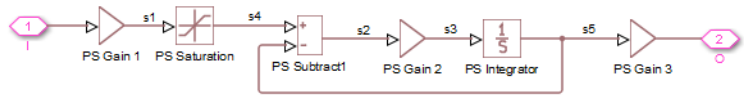
Library

Valve Actuators

Description

The Valve Actuator block represents a simplified model of a valve actuator, which is especially suitable for real-time and HIL (hardware-in-the-loop) simulation. Its functionality is similar to the 2-Position Valve Actuator and 3-Position Valve Actuator blocks, but its characterization requires fewer parameters because it is built of components that are practically linear.

The Valve Actuator block is built using the blocks from the Simscape Physical Signals library. Both the input and the output of the block are physical signals. The block diagram of the model is shown in the following figure.



The gain of the PS Gain 1 block establishes the steady-state relationship between the input and output signals. The PS Saturation block keeps the output signal within the +/– valve stroke range. The gain of the PS Gain 2 block is 1/**Time constant**, which makes it possible to reproduce the first-order lag by combining this block with the PS Subtract and PS Integrator blocks.

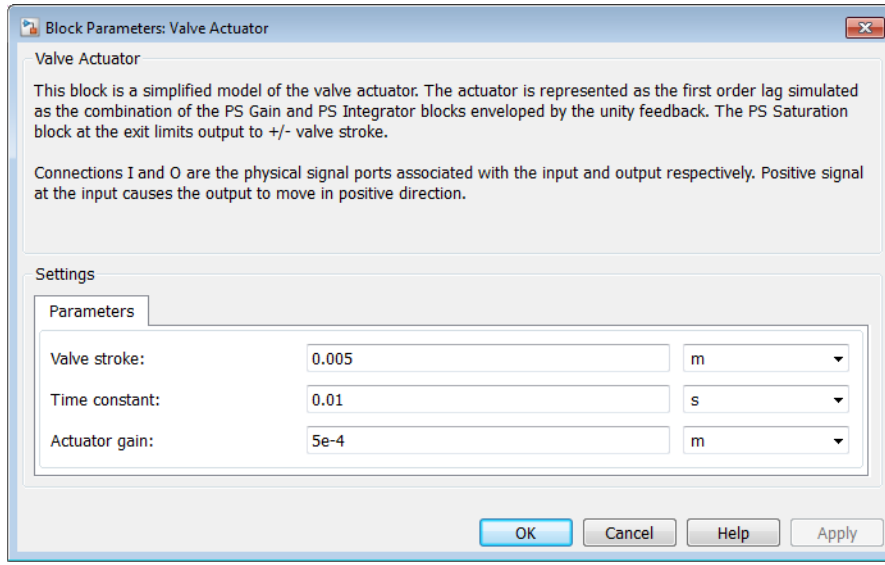
The transfer function of the lag is

$$H(s) = \frac{1}{Ts+1}$$

where T is the time constant. The gain of the PS Gain 3 block is set to 1.

Note This block is constructed out of blocks from the Simscape Physical Signals library (such as PS Add, PS Gain, and so on). Currently, the blocks in the Physical Signals library do not support unit propagation and checking. For more information, see “How to Work with Physical Units” (Simscape).

Dialog Box and Parameters



Valve stroke

The desired valve stroke, which defines the saturation level of the PS Saturation block in the actuator model. The output of the actuator is kept within the +/– valve stroke range. The default value is `0.005` m.

Time constant

Time constant of the first-order lag. The **Gain** parameter of the PS Gain 2 block is equal to 1/**Time constant**. The default value is `0.01` s.

Actuator gain

Gain of the PS Gain 1 block in the block diagram of the actuator model. The default value is `5e-4` m.

Ports

The block has one physical signal input port and one physical signal output port.

See Also

4-Way Ideal Valve | Proportional and Servo-Valve Actuator

Introduced in R2010a

Valve Hydraulic Force

Axial hydraulic static force exerted on valve

Library

Valve Forces

Description

The Valve Hydraulic Force block simulates axial hydraulic static force exerted on a valve by fluid flowing through the orifice. The relationship between the valve opening, the pressure drop, and the force is provided as a two-dimensional table, which is processed by the PS Lookup Table (2D) block. The table can be obtained experimentally or analytically and can represent both the hydraulic static axial force and pressure forces. The force matrix must be rectangular and contain as many rows as there are pressure differential measurements and as many columns as there are valve openings. The pressure differential and opening vectors must be arranged in strictly ascending order and cover the whole range of valve operation. Connect the block in parallel with the orifice whose flow induces the force.

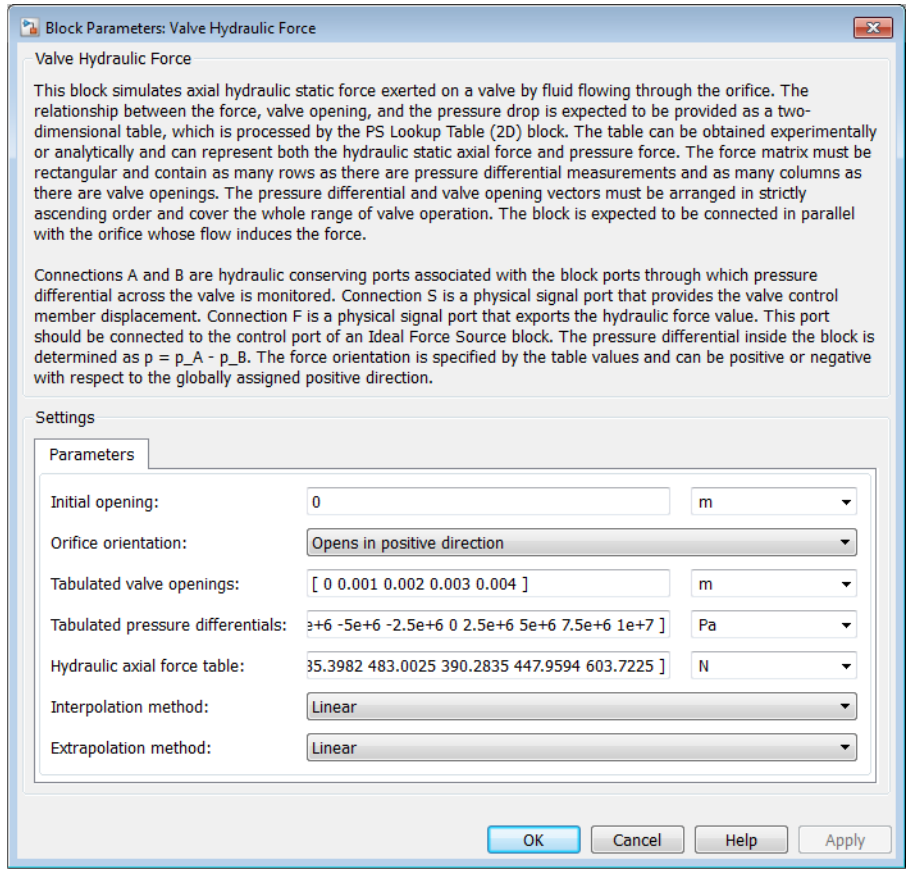
Connections A and B are hydraulic conserving ports that should be connected to the valve block ports in such a way as to monitor the pressure differential across the valve. Connection S is a physical signal port that provides the valve control member displacement. Connection F is a physical signal port that outputs the hydraulic axial force value. This port should be connected to the control port of an Ideal Force Source block.

The pressure differential inside the block is determined as $\Delta p = p_A - p_B$, . The force orientation is specified by the table values and can be positive or negative with respect to the globally assigned positive direction, depending on the value of the **Orifice orientation** parameter.

Basic Assumptions and Limitations

No transient effects can be simulated.

Dialog Box and Parameters



Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The default value is `0`.

Orifice orientation

The parameter is introduced to specify the effect of the valve opening on the valve force. The parameter can be set to one of two options: `Opens in positive direction` or `Opens in negative direction`. The value `Opens in positive direction` specifies an orifice that opens when the valve is shifted in the globally assigned positive direction. The default value is `Opens in positive direction`.

Tabulated valve openings

Specify the vector of input values for valve openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[0,1e-3,2e-3,3e-3,4e-3]`. The **Tabulated valve openings** values will be used together with **Tabulated pressure differentials** for two-dimensional table lookup in the **Hydraulic axial force table**.

Tabulated pressure differentials

Specify the vector of input values for pressure differentials as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-100e5,-75e5,-50e5,-25e5,0,25e5,50e5,75e5,100e5]`.

Hydraulic axial force table

Specify the hydraulic axial force as an `m`-by-`n` matrix, where `m` is the number of valve openings and `n` is the number of pressure differentials. Each value in the matrix specifies an axial force corresponding to a specific combination of valve opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in N, are:

```
       [0, -127.3576,  -27.8944,  227.2513,  575.3104; ...
        0,  -95.5182,  -20.9208,  170.4385,  431.4828; ...
        0,  -63.6788,  -13.9472,  113.6256,  287.6552; ...
        0,  -31.8394,   -6.9736,   56.8128,  143.8276; ...
        0,         0,         0,         0,         0; ...
 196.3495,  120.7506,   97.5709,  111.9898,  150.9306; ...
 392.6991,  241.5013,  195.1418,  223.9797,  301.8613; ...
 589.0486,  362.2519,  292.7126,  335.9695,  452.7919; ...
 785.3982,  483.0025,  390.2835,  447.9594,  603.7225]
```

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous surface with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (2D) block reference page.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (2D) block reference page.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Orifice orientation**
- **Interpolation method**
- **Extrapolation method**

All other block parameters are available for modification.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with a valve port.

B

Hydraulic conserving port associated with another valve port to monitor the pressure differential.

S

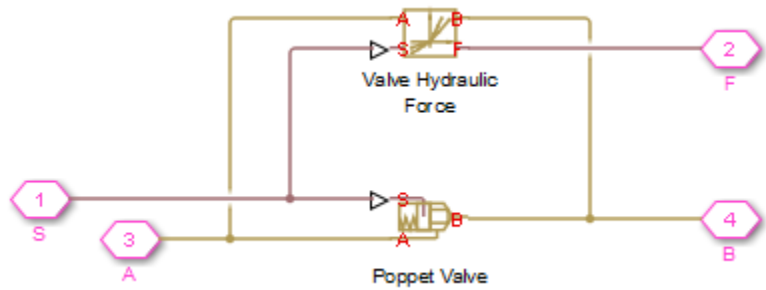
Physical signal port that provides the valve control member displacement.

F

Physical signal port that outputs hydraulic axial force.

Examples

The following example shows a model of a poppet valve built of a Poppet Valve block and a Valve Hydraulic Force block. The Valve Hydraulic Force block is connected in parallel and provides tabulated data to compute hydraulic force acting on the valve. The force value is exported through the F port.



See Also

Spool Orifice Hydraulic Force

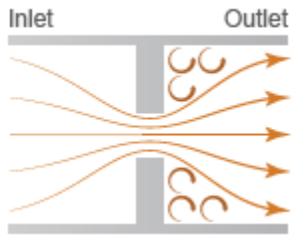
Introduced in R2007a

Variable Area Orifice (TL)

Local flow restriction with a variable cross-sectional area
Library: Simscape / Fluids / Thermal Liquid / Valves & Orifices

Description

The Variable Area Orifice (TL) block models the flow through a local restriction with variable opening area. The orifice contains a control member—such as a ball, spool, or diaphragm—which determines by its displacement the instantaneous opening area. Elements such as this are characteristic of valves and are, in the Thermal Liquid library, the foundation upon which all directional valve blocks are based. See, for example, the 2-Way Directional Valve (TL) block. Use this block to create a custom component with variable orifices if such is not provided in the Thermal Liquid library.



The orifice is assumed to consist of a contraction followed by a sudden expansion in flow area. The contraction causes the flow rate to rise and the pressure to drop. The expansion allows the pressure to recover, though only in part: past the vena contracta, where the flow is at its narrowest, the flow generally separates from the wall, causing it to lose some energy. The extent of the pressure recovery depends on the discharge coefficient of the orifice and on the ratio of the orifice and port areas. Set the **Pressure recovery** to `Off` to ignore this effect if necessary.

The effect that the motion of the control member has on the opening area of the orifice depends on the setting of the **Opening orientation** block parameter. In the default setting of `Positive`, the orifice (if within its opening range) opens when the control

member moves in the positive direction. In the alternate setting of `Negative`, the orifice opens with motion in the negative direction.

Orifice Positions

The orifice is continuously variable. It shifts smoothly between positions, of which it has two. One—the *normal* position—is that to which the orifice reverts when its control signal falls to zero. Unless a control member offset has been specified, the **A**–**B** orifice is always fully closed in this position. Another—the *working* position—is that to which the orifice moves when its control signal rises to a maximum. The orifice is generally fully open in this position. Note that whether the orifice is in fact open and how open it is both depend on the value of the control member offset.

Orifice Opening

Which position the orifice is in depends on the control member coordinate—a length that, in the valve blocks based on this orifice model, is often referred to as the *orifice opening*. This variable is calculated during simulation from the control member offset, specified via the block parameter of the same name, and from the control member displacement, a variable obtained from the physical signal specified at port **S**:

$$h = h_0 + \delta x,$$

where:

- h is the **A**–**B** orifice opening.
- h_0 is the **A**–**B** opening offset.
- δ is the orifice orientation, `+1` if `Positive`, `-1` if `Negative`.
- x is the control member displacement.

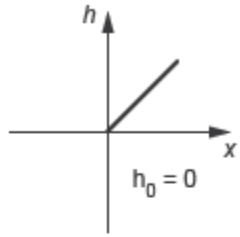
A control member displacement of zero corresponds to a valve that is in its normal position. The orifice begins to open when the orifice opening (h) rises above zero and it continues to open until the orifice opening is at a maximum value. This maximum is obtained from the **Maximum control displacement** block parameter, in the linear orifice parameterization, or from the specified data vectors, in the tabulated orifice parameterizations.

Opening Offsets

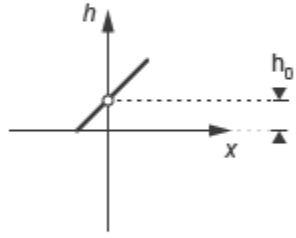
The orifice is by default configured so that it is fully closed when the control member displacement is zero. Such an orifice, when it represents a valve, is often described as being zero-lapped. It is possible, by applying an offset to the control member, to model an orifice that is *underlapped*—that is, partially open when in the normal position. The orifice can also be *overlapped*—fully closed over a range of control member displacements extending past the normal closed position.

The figure graphs the orifice opening—*h(x)*—in the cases of zero-lapped (**I**), underlapped (**II**), and overlapped (**III**) orifices. The opening offset—hs_0—is zero in the first case, greater than zero in the second, and smaller than zero in the third. The control member must move right of its normal position (in the positive direction along the *x*-axis) for the overlapped orifice to crack open; it must move left of its normal position for the underlapped orifice to shut tight.

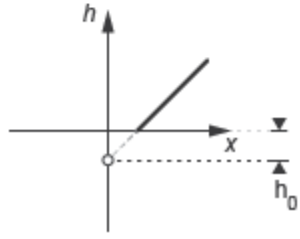
I.



II.



III.



Opening Characteristics

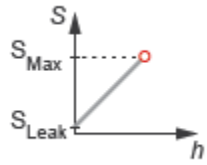
The orifice opening serves during simulation to calculate the mass flow rate through the orifice. The calculation can be a direct mapping from opening to flow rate or an indirect conversion, first from opening to orifice area and then from orifice area to mass flow rate. The calculation, and the data required for it, depend on the setting of the **Valve parameterization** block parameter:

- `Linear area-opening relationship` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve.

The opening area is assumed to vary linearly with the control member position. The slope of the linear expression is determined from the **Maximum valve opening** and **Maximum opening area** block parameters:

$$S_{\mathrm{Lin}} = \frac{S_{\mathrm{Max}}}{h_{\mathrm{Max}}} h,$$

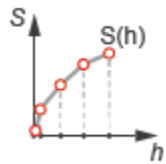
where S_{Lin} is the linear form of the opening area, S_{Max} is the value of the **Maximum orifice area** block parameter, h_{Max} is the value of the **Maximum control displacement** block parameter. This expression is reformulated as a piecewise conditional expression so as to saturate the opening area at a small leakage value and ensure that transitions to the normal and working positions are smooth.



- `Tabulated data - Area vs. opening` — Calculate the valve opening area from the control member position and from it obtain the mass flow rate through the valve. The opening area can vary nonlinearly with the control member position. The relationship between the two is given by the tabulated data in the **Valve opening vector** and **Opening area vector** block parameters:

  $$S_{\mathrm{Tab}} = S(h),$$

  where S_{Tab} is the tabulated form of the opening area, a function of the orifice opening, h.

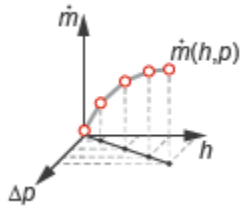


- `Tabulated data - Mass flow rate vs. opening and pressure drop` — Calculate the mass flow rate directly from the control member position and the

pressure drop across the valve. The relationship between the three variables can be nonlinear and it is given by the tabulated data in the **Valve opening vector**, **Pressure drop vector**, and **Mass flow rate table** block parameters:

$$\dot{m}_{\mathrm{Tab}} = \frac{\rho_{\mathrm{Ref}}}{\rho_{\mathrm{Avg}}}\dot{m}(h, \Delta p),$$

where $\dot{m}$ is the tabulated form of the mass flow rate, a function of the orifice opening, h, and of the pressure drop across the orifice, Δp. The mass flow rate is adjusted for temperature and pressure by the ratio $\rho_{\mathrm{Ref}}/\rho_{\mathrm{Avg}}$, where ρ is the fluid density at some reference temperature and pressure (subscript `Ref`) or at the averages of those variables within the orifice.



Numerical Smoothing

To ensure adequate simulation performance, the orifice opening area is smoothed over two small regions of the orifice opening, one near the fully closed state, the other near the fully open state. The smoothing is accomplished by means of polynomial expressions (to be incorporated into the final form of the opening area expression):

$$\lambda_{\mathrm{Min}} = 3\Delta h^{*}_{\mathrm{Min}} - 2\Delta h^{*3}_{\mathrm{Min}} \quad \text{and} \quad \lambda_{\mathrm{Max}} = 3\Delta h^{*}_{\mathrm{Max}} - 2\Delta h^{*3}_{\mathrm{Max}},$$

where λ is the smoothing factor applied at the minimum (subscript `Min`) and maximum (subscript `Max`) portions of the opening area expression. The smoothing factors are calculated as:

$$\Delta h_{\mathrm{Min}} = \frac{h - h_{\mathrm{Min}}}{\Delta h_{\mathrm{Smooth}}} \quad \text{and} \quad \Delta h_{\mathrm{Max}} = \frac{h - (h_{\mathrm{Max}} - \Delta h_{\mathrm{Smooth}})}{\Delta h_{\mathrm{Smooth}}},$$

where h_{Min} is the minimum orifice opening and Δh_{Smooth} is the range of orifice openings over which to smooth the linear form of the opening area. The value of S_{Min} is calculated as:

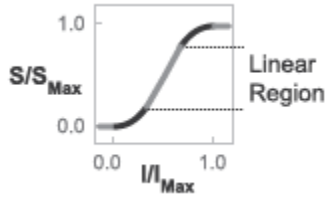
$$h_{\text{Min}} = h_{\text{Max}} \frac{S_{\text{Leak}}}{S_{\text{Max}}},$$

where S_{Leak} is the value of the **Leakage area** block parameter. The value of S_{Smooth} is calculated as:

$$\Delta h_{\text{Smooth}} = f_{\text{Smooth}} \frac{h_{\text{Max}} - h_{\text{Min}}}{2},$$

where f_{Smooth} is the value of the **Smoothing factor** block parameter—a fraction between `0` and `1`, with `0` indicating zero smoothing and `1` maximum smoothing. The final, smoothed, orifice opening area is given by the piecewise expression:

$$S_{\text{Smooth}} = \begin{cases} S_{\text{Leak}}, & \text{if } h \le h_{\text{Min}} \\ S_{\text{Leak}}\left(1-\lambda_{\text{Min}}\right) + S_{\text{Leak}}\lambda_{\text{Min}}, & \text{if } h < h_{\text{Min}} + \Delta h_{\text{Smooth}} \\ S_{\text{Lin}}, & \text{if } h \le h_{\text{Max}} - \Delta h_{\text{Smooth}} \\ S_{\text{Lin}}\left(1-\lambda_{\text{Max}}\right) + S_{\text{Max}}\lambda_{\text{Max}}, & \text{if } h < h_{\text{Max}} \\ S_{\text{Max}}, & \text{if } h \ge h_{\text{Max}} \end{cases}.$$



Orifice Area Smoothing

Leakage Flow

The primary purpose of the leakage flow rate of a closed orifice is to ensure that at no time a portion of the thermal liquid network becomes isolated from the remainder of the

model. Such isolated portions reduce the numerical robustness of the model and can slow down simulation or cause it to fail. Leakage flow is generally present in real orifices but in a model its exact value is less important than its being a small number greater than zero. The leakage flow rate is determined from the **Leakage area** block parameter.

Mass Balance

The volume of fluid inside the orifice, and therefore the mass of the same, is assumed to be very small and it is, for modeling purposes, ignored. As a result, no amount of fluid can accumulate there. By the principle of conservation of mass, the mass flow rate into the orifice through one port must therefore equal that out of the orifice through the other port:

$$\dot{m}_A + \dot{m}_B = 0,$$

where $\dot{m}$ is defined as the mass flow rate *into* the orifice through the port indicated by the subscript (**A** or **B**).

Momentum Balance

The causes of the pressure losses incurred in the orifice are ignored in the block. Whatever their natures—sudden area changes, flow passage contortions—only their cumulative effect is considered during simulation. This effect is captured in the block by the discharge coefficient, a measure of the mass flow rate through the orifice relative to the theoretical value that it would have in an ideal orifice. Expressing the momentum balance in the orifice in terms of the pressure drop induced in the flow:

$$p_{\mathrm{A}} - p_{\mathrm{B}} = \frac{\dot{m}_{\mathrm{Avg}}\sqrt{\dot{m}_{\mathrm{Avg}}^2 + \dot{m}_{\mathrm{Crit}}^2}}{2\rho_{\mathrm{Avg}} C_{\mathrm{D}}^2 S_{\mathrm{Smooth}}^2}\left[1 - \left(\frac{S_{\mathrm{Smooth}}}{S_{\mathrm{Lin}}}\right)^2\right]\xi_{\mathrm{p}},$$

where C_{D} is the discharge coefficient, and ξ_{p} is the pressure drop ratio—a measure of the effect impressed by the pressure recovery that in real orifices occurs between the vena contracta (the point at which the flow is at its narrowest) and the outlet, assumed to be a small distance away. The subscript `Avg` denotes an average of the values at the thermal liquid ports. The critical mass flow rate $\dot{m}_{\mathrm{Crit}}$ is calculated from the critical Reynolds number—that at which the flow in the orifice is assumed to transition from laminar to turbulent:

$$\dot{m}_{\mathrm{Crit}} = \mathrm{Re}_{\mathrm{Crit}} \mu_{\mathrm{Avg}} \sqrt{\frac{\pi}{4} S_{\mathrm{Lin}}},$$

where μ denotes dynamic viscosity. The value of the pressure ratio depends on the setting of the **Pressure recovery** block parameter. In the default setting of `Off`:

$$\xi_{\mathrm{p}} = 1.$$

If `On` is selected instead:

$$\xi_{\mathrm{p}} = \frac{\sqrt{1 - \left(\frac{S_{\mathrm{Smooth}}}{S_{\mathrm{Lin}}}\right)^2 \left(1 - C_{\mathrm{D}}^2\right)} - C_{\mathrm{D}} \frac{S_{\mathrm{Smooth}}}{S_{\mathrm{Lin}}}}{\sqrt{1 - \left(\frac{S_{\mathrm{Smooth}}}{S_{\mathrm{Lin}}}\right)^2 \left(1 - C_{\mathrm{D}}^2\right)} + C_{\mathrm{D}} \frac{S_{\mathrm{Smooth}}}{S_{\mathrm{Lin}}}}.$$

Energy Balance

The orifice is modeled as an adiabatic component. No heat exchange can occur between the fluid and the wall that surrounds it. No work is done on or by the fluid as it traverses from inlet to outlet. With these assumptions, energy can flow by advection only, through ports **A** and **B**. By the principle of conservation of energy, the sum of the port energy flows must always equal zero:

$$\phi_{\mathrm{A}} + \phi_{\mathrm{B}} = 0,$$

where ϕ is defined as the energy flow rate *into* the orifice through one of the ports (**A** or **B**).

Ports

Input

S — Control member displacement, `unitless`

physical signal

Instantaneous displacement of the valve control member.

Conserving

`A` — Flow passage
thermal liquid

Opening through which the flow can enter or exit the valve.

`B` — Flow passage
thermal liquid

Opening through which the flow can enter or exit the valve.

Parameters

`Orifice parameterization` — Method by which to model the opening characteristics of the orifice
`Linear area-opening relationship` (default) | `Tabulated data - Area vs. opening` | `Tabulated data - Mass flow rate vs. opening and pressure drop`

Method by which to model the opening characteristics of the orifice. The default setting prescribes a linear relationship between the orifice opening area and the orifice opening. The alternative settings allow for a general, nonlinear relationship to be specified in tabulated form, in one case between the opening area and the orifice opening, in the other case between the mass flow rate and both the orifice opening and the pressure drop between the ports.

`Maximum valve opening` — Orifice opening at which the opening area is at a maximum
`5e-3 m` (default) | positive scalar in units of length

Orifice opening at which the orifice is fully open and its opening area is therefore at a maximum. This parameter is used to calculate the slope of the linear expression relating the opening area to the orifice opening.

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Linear area-opening relationship`.

`Maximum orifice area` — Opening area of the orifice in the fully open position

`1e-4 m^2` (default) | positive scalar in units of area

Opening area of the orifice in the fully open position, when the orifice opening is that specified in the **maximum valve opening** block parameter. This parameter is used to calculate the slope of the linear expression relating the opening area to the orifice opening.

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Linear area-opening relationship`.

`Leakage area` — Opening area of the orifice in the maximally closed position

`1e-10 m^2` (default) | positive scalar in units of area

Opening area of the orifice in the fully closed position, when only internal leakage between its ports remains. This parameter serves primarily to ensure that closure of the orifice does not cause portions of the thermal liquid network to become isolated. The exact value specified here is less important than its being a small number greater than zero.

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Linear area-opening relationship`.

`Smoothing factor` — Measure of the amount of smoothing to apply to the opening area function

`0.01` (default) | positive unitless scalar

Measure of the amount of smoothing to apply to the opening area function. This parameter determines the widths of the regions to be smoothed, one being at the fully open position, the other at the fully closed position. The smoothing superposes on the linear opening area function two nonlinear segments, one for each region of smoothing. The greater the value specified, the greater the smoothing and the broader the nonlinear segments.

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Linear area-opening relationship`.

`Opening vector` — Vector of orifice openings at which to tabulate the orifice opening areas

`[-0.002, 0, 0.002, 0.005, 0.015] m` (default) | vector of positive numbers in units of length

Vector of orifice openings at which to specify—dependent on the valve parameterization—the opening area of the orifice or its mass flow rate. The vector elements must increase monotonically from left to right. This vector must be equal in size to that specified in the **Opening area vector** block parameter or to the number of rows in the **Mass flow rate table** block parameter.

This data serves to construct a one-way lookup table by which to determine, from the orifice opening, the opening area of the orifice or a two-way lookup table by which to determine, from the orifice opening and pressure drop, the mass flow rate of the orifice. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Opening area vector` — Vector of opening areas at the specified orifice opening breakpoints

`[1e-09, 2.0352e-07, 4.0736e-05, 0.00011438, 0.00034356] m^2` (default) | vector of positive numbers in units of area

Vector of opening areas corresponding to the breakpoints defined in the **Opening vector** block parameter. The vector elements must increase monotonically from left to right (with increasing values of the orifice opening). This vector must be equal in size to the number of orifice opening breakpoints.

This data serves to construct a one-way lookup table by which to determine from the orifice opening the opening area of the orifice. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Tabulated data - Area vs. opening`.

`Pressure drop vector` — Vector of pressure drops at which to tabulate the mass flow rates

`[-0.1, 0.3, 0.5, 0.7] MPa` (default) | vector in units of pressure

Vector of pressure differentials from port **A** to port **B** at which to specify the mass flow rate of the orifice. The vector elements must increase monotonically from left to right. This vector must be equal in size to the number of columns in the **Mass flow rate table** block parameter.

This data serves to construct a two-way lookup table by which to determine, from the orifice opening and pressure drop, the opening area of the orifice. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range).

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Mass flow rate table` — Matrix of mass flow rates at the specified orifice opening and pressure drop breakpoints

`[-1e-05, 1.7e-05, 2e-05, 2.6e-05; -.002, .0035, .0045, .0053; -.4, .7, .9, 1.06; -1.13, 1.96, 2.5, 3; -3.45, 6, 7.7, 9.13] kg/s` (default) | matrix in units of mass/time

Matrix of mass flow rates corresponding to the breakpoints defined in the **Opening vector** and **Pressure drop vector** block parameters. The orifice opening increases from row to row from top to bottom. The pressure drop increases from column to column from left to right. The mass flow rate must increase monotonically in the same directions (with increasing control member position and increasing pressure drop).

This data serves to construct a two-way lookup table by which to determine, from the orifice opening and pressure drop, the opening area of the orifice. Data is handled with linear interpolation (within the tabulated data range) and nearest-neighbor extrapolation (outside of the data range). Ensure that the number of rows is equal to the size of the **Opening area vector** block parameter and that the number of columns is equal to the size of the **Pressure drop vector** block parameter.

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow temperature` — Inlet temperature at which to specify the tabulated data

`293.15 K` (default) | positive scalar in units of temperature

Nominal inlet temperature, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the temperature measured during simulation.

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Reference inflow pressure` — Inlet pressure at which to specify the tabulated data

`0.101325 MPa` (default) | positive scalar in units of pressure

Nominal inlet pressure, with reference to absolute zero, at which to specify the tabulated data. This parameter is used to adjust the mass flow rate according to the pressure measured during simulation.

Dependencies

This parameter is active when the **Orifice parameterization** block parameter is set to `Tabulated data - Mass flow rate vs. opening and pressure drop`.

`Control member offset` — Distance by which to displace the control member in the normal orifice position

`0 m` (default) | positive scalar in units of length

Offset between the control member and the location at which, in the normal orifice position, it would completely cover the orifice. Specify a positive offset to model an underlapped orifice or a negative offset to model an overlapped orifice. For detail on how the opening offsets impact the block calculations, see the block description.

`Cross-sectional area at ports A and B` — Flow area at the thermal liquid ports

`0.01 m` (default) | positive scalar in units of area

Area normal to the flow path at each port. The ports are assumed to be equal in size. The flow area specified here should match those of the inlets of those components to which the orifice connects.

`Characteristic longitudinal length` — Measure of the length of the flow path through the orifice
`0.1` (default) | positive scalar in units of length

Average distance traversed by the fluid as it moves from inlet to outlet. This distance is used in the calculation of the internal thermal conduction that occurs between the two ports (as part of the smoothed upwind energy scheme employed in the thermal liquid domain).

`Discharge coefficient` — Empirical factor defined as the ratio of actual to ideal mass flow rates
`0.7` (default) | positive unitless scalar

Ratio of the actual flow rate through the orifice to the theoretical value that it would have in an ideal valve. This semi-empirical parameter measures the flow allowed through the orifice: the greater its value, the greater the flow rate. Refer to the valve data sheet, if available, for this parameter.

`Critical Reynolds number` — Reynolds number at the boundary between laminar and turbulent flow regimes
12 (default) | positive unitless scalar

Reynolds number at which the flow is assumed to transition between laminar and turbulent regimes.

References

[1] *Measurement of fluid flow by means of pressure differential devices inserted in circular cross-section conduits running full — Part 2: Orifice plates (ISO 5167–2:2003)*. 2003.

See Also

2-Way Directional Valve (TL) | 3-Way Directional Valve (TL) | 4-Way Directional Valve (TL) | Check Valve (TL)

Introduced in R2016a

Variable Orifice

Generic hydraulic variable orifice

Library

Orifices

Description

The block represents a variable orifice of any type as a data-sheet-based model. Depending on data listed in the manufacturer's catalogs or data sheets for your particular orifice, you can choose one of the following model parameterization options:

- `By maximum area and opening` — Use this option if the data sheet provides only the orifice maximum area and the control member maximum stroke.
- `By area vs. opening table` — Use this option if the catalog or data sheet provides a table of the orifice passage area based on the control member displacement $A=A(h)$.
- `By pressure-flow characteristic` — Use this option if the catalog or data sheet provides a two-dimensional table of the pressure-flow characteristics $q=q(p,h)$.

In the first case, the passage area is assumed to be linearly dependent on the control member displacement, that is, the orifice is assumed to be closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. In the second case, the passage area is determined by one-dimensional interpolation from the table $A=A(h)$. In both cases, a small leakage area is assumed to exist even after the orifice is completely closed. Physically, it represents a possible clearance in the closed valve, but the main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. An isolated or “hanging” part of the system could affect computational efficiency and even cause failure of computation.

In the first and second cases, the flow rate is computed according to the following equations:

$$q = C_D \cdot A(h)\sqrt{\frac{2}{\rho}}\frac{\Delta p}{\left(\Delta p^2 + p_{\mathrm{Cr}}^2\right)^{1/4}},$$

$$\Delta p = p_{\mathrm{A}} - p_{\mathrm{B}},$$

$$h = x_0 + x \bullet or$$

$$A(h) = \begin{cases} h \bullet A_{\max} / h_{\max} + A_{leak} & \text{for } h > 0 \\ A_{leak} & \text{for } h <= 0 \end{cases}$$

where

q	Flow rate
p	Pressure differential
p_{A}, p_{B}	Gauge pressures at the block terminals
C_{D}	Flow discharge coefficient
$A(h)$	Instantaneous orifice passage area
$A_{\max}$	Orifice maximum area
$h_{\max}$	Control member maximum displacement
x_0	Initial opening
x	Control member displacement from initial position
h	Orifice opening
or	Orifice orientation indicator. The variable assumes +1 value if the control member displacement in the globally assigned positive direction opens the orifice, and –1 if positive motion decreases the opening.
ρ	Fluid density
A_{leak}	Closed orifice leakage area
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

  $$p_{avg} = (p_A + p_B)/2$$

  where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa
B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = \frac{\rho}{2}\left(\frac{\mathrm{Re}_{cr}\cdot\nu}{C_D\cdot D_H}\right)^2$$

  $$D_H = \sqrt{\frac{4A}{\pi}}$$

  where

D_H	Instantaneous orifice hydraulic diameter
ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

In the third case, when an orifice is defined by its pressure-flow characteristics, the flow rate is determined by two-dimensional interpolation. In this case, neither flow regime nor leakage flow rate is taken into account, because these features are assumed to be introduced through the tabulated data. Pressure-flow characteristics are specified with three data sets: array of orifice openings, array of pressure differentials across the orifice, and matrix of flow rate values. Each value of a flow rate corresponds to a specific combination of an opening and pressure differential.

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B and the pressure differential is determined as

$\Delta p = p_A - p_B$, . Positive signal at the physical signal port S opens or closes the orifice depending on the value of the orifice orientation indicator.

Basic Assumptions and Limitations

- Fluid inertia is not taken into account.
- For orifices specified by pressure-flow characteristics (the third parameterization option), the model does not explicitly account for the flow regime or leakage flow rate, because the tabulated data is assumed to account for these characteristics.

Parameters

Model parameterization

Select one of the following methods for specifying the orifice:

- `By maximum area and opening` — Provide values for the maximum orifice area and the maximum orifice opening. The passage area is linearly dependent on the control member displacement, that is, the orifice is closed at the initial position of the control member (zero displacement), and the maximum opening takes place at the maximum displacement. This is the default method.
- `By area vs. opening table` — Provide tabulated data of orifice openings and corresponding orifice areas. The passage area is determined by one-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.
- `By pressure-flow characteristic` — Provide tabulated data of orifice openings, pressure differentials, and corresponding flow rates. The flow rate is determined by two-dimensional table lookup. You have a choice of two interpolation methods and two extrapolation methods.

Orifice maximum area

Specify the area of a fully opened orifice. The parameter value must be greater than zero. The default value is `5e-5` m^2. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Orifice maximum opening

Specify the maximum displacement of the control member. The parameter value must be greater than zero. The default value is `5e-4` m. This parameter is used if **Model parameterization** is set to `By maximum area and opening`.

Orifice opening vector, s

Specify the vector of input values for orifice openings as a one-dimensional array. The input values vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in meters, are `[-0.002 0 0.002 0.005 0.015]`. If **Model parameterization** is set to `By area vs. opening table`, the **Tabulated orifice openings** values will be used together with **Tabulated orifice area** values for one-dimensional table lookup. If **Model parameterization** is set to `By pressure-flow characteristic`, the **Tabulated orifice openings** values will be used together with **Tabulated pressure differentials** and **Tabulated flow rates** for two-dimensional table lookup.

Orifice area vector

Specify the vector of orifice areas as a one-dimensional array. The vector must be of the same size as the orifice openings vector. All the values must be positive. The default values, in m^2, are `[1e-09 2.0352e-07 4.0736e-05 0.00011438 0.00034356]`. This parameter is used if **Model parameterization** is set to `By area vs. opening table`.

Pressure differential vector, dp

Specify the pressure differential vector as a one-dimensional array. The vector must be strictly increasing. The values can be nonuniformly spaced. The minimum number of values depends on the interpolation method: you must provide at least two values for linear interpolation, at least three values for smooth interpolation. The default values, in Pa, are `[-1e+07 -5e+06 -2e+06 2e+06 5e+06 1e+07]`. This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Volumetric flow rate table, q(s,dp)

Specify the flow rates as an `m`-by-`n` matrix, where `m` is the number of orifice openings and `n` is the number of pressure differentials. Each value in the matrix specifies flow rate taking place at a specific combination of orifice opening and pressure differential. The matrix size must match the dimensions defined by the input vectors. The default values, in m^3/s, are:

```
[-1e-07 -7.0711e-08 -4.4721e-08 4.4721e-08 7.0711e-08 1e-07;
 -2.0352e-05 -1.4391e-05 -9.1017e-06 9.1017e-06 1.4391e-05 2.0352e-05;
 -0.0040736 -0.0028805 -0.0018218 0.0018218 0.0028805 0.0040736;
 -0.011438 -0.0080879 -0.0051152 0.0051152 0.0080879 0.011438;
 -0.034356 -0.024293 -0.015364 0.015364 0.024293 0.034356;]
```

This parameter is used if **Model parameterization** is set to `By pressure-flow characteristic`.

Interpolation method

Select one of the following interpolation methods for approximating the output value when the input value is between two consecutive grid points:

- `Linear` — Select this option to get the best performance.
- `Smooth` — Select this option to produce a continuous curve (`By area vs. opening table`) or surface (`By pressure-flow characteristic`) with continuous first-order derivatives.

For more information on interpolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Extrapolation method

Select one of the following extrapolation methods for determining the output value when the input value is outside the range specified in the argument list:

- `Linear` — Select this option to produce a curve or surface with continuous first-order derivatives in the extrapolation region and at the boundary with the interpolation region.
- `Nearest` — Select this option to produce an extrapolation that does not go above the highest point in the data or below the lowest point in the data.

For more information on extrapolation algorithms, see the PS Lookup Table (1D) and PS Lookup Table (2D) block reference pages.

Orifice orientation

The parameter is introduced to specify the effect of the orifice control member motion on the valve opening. The parameter can be set to one of two options: `Opens in positive direction` or `Opens in negative direction`. The value `Opens in positive direction` specifies an orifice whose control member opens the valve when it is shifted in the globally assigned positive direction. The parameter is extremely useful for building a multi-orifice valve with all the orifices being controlled by the same spool. The default value is `Opens in positive direction`.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Initial opening

Orifice initial opening. The parameter can be positive (underlapped orifice), negative (overlapped orifice), or equal to zero for zero lap configuration. The value of initial opening does not depend on the orifice orientation. The default value is `0`.

Leakage area

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the valve is completely closed. The parameter value must be greater than 0. The default value is `1e-12` m^2.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Variables Tab

Volumetric flow rate

Flow rate specified as the fluid volume per unit time at time zero. The value entered serves as a guide during model compilation. The actual flow rate can differ if needed

to satisfy all model constraints. Set the **Priority** level to `High` to prioritize the specified value during model assembly.

Pressure drop

Pressure change from port A to port B at time zero. The value entered serves as a guide during model compilation. The actual flow rate can differ if needed to satisfy all model constraints. Set the **Priority** level to `High` to prioritize the specified value during model assembly.

Restricted Parameters

When your model is in Restricted editing mode, you cannot modify the following parameters:

- **Model parameterization**
- **Orifice orientation**
- **Interpolation method**
- **Extrapolation method**
- **Laminar transition specification**

All other block parameters are available for modification. The actual set of modifiable block parameters depends on the value of the **Model parameterization** parameter at the time the model entered Restricted mode.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port to control spool displacement.

The flow rate is positive if fluid flows from port A to port B. Positive signal at the physical signal port S opens or closes the orifice depending on the value of the parameter **Orifice orientation**.

Examples

The Hydraulic Flapper-Nozzle Amplifier example illustrates the use of the Variable Orifice block in hydraulic systems.

See Also

Annular Orifice | Constant Area Hydraulic Orifice | Fixed Orifice | Orifice with Variable Area Round Holes | Orifice with Variable Area Slot | PS Lookup Table (1D) | PS Lookup Table (2D) | Variable Area Hydraulic Orifice

Introduced in R2006a

Variable Orifice Between Round Holes

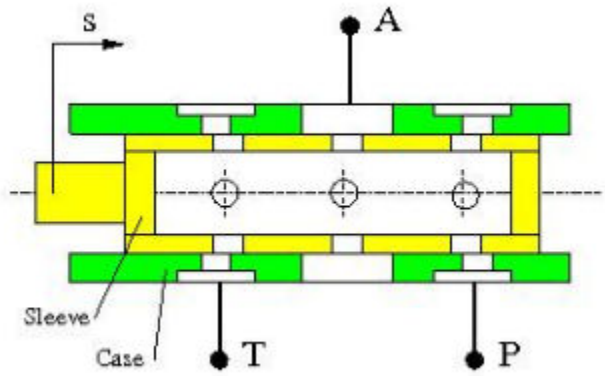
Hydraulic variable orifice at intersection of two holes

Library

Orifices

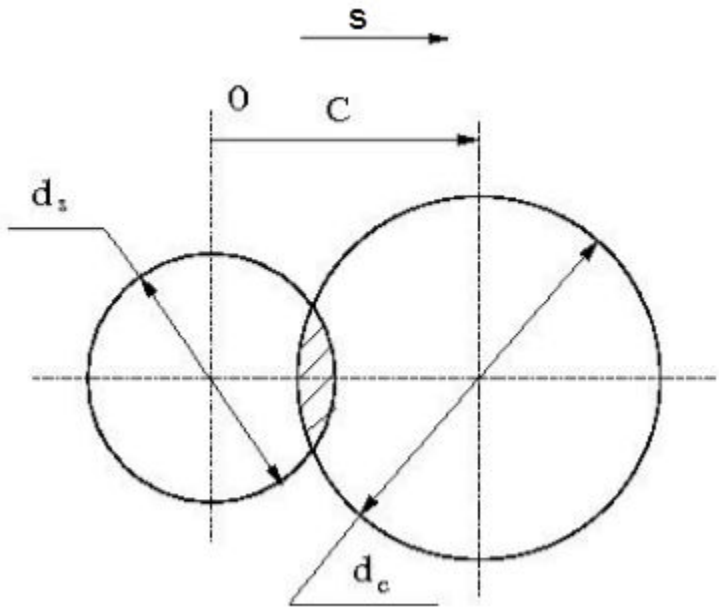
Description

The Variable Orifice Between Round Holes block models a variable orifice created by two interacting round holes. These holes can have different diameters. One hole is located in the sleeve, while the other is drilled in the case, with the sleeve sliding along the case. Such a configuration is frequently seen in cartridge valves, as shown in this 3-way valve schematic.



The block can contain multiple identical interacting pairs of holes. The following schematic shows the calculation diagram for one such pair of round holes, where

s	Sleeve displacement from initial position
c	Distance between hole centers
d_s	Sleeve hole diameter
d_c	Case hole diameter



The flow rate through the orifice is proportional to the orifice area and to the pressure differential across the orifice, according to these equations:

$$A = \begin{cases} A_{leak} & \text{for c} \geq (r+R) \\ A_{leak} + r^2 \cdot a\cos\left(\frac{c^2+r^2-R^2}{2cr}\right) + R^2 \cdot a\cos\left(\frac{c^2-r^2+R^2}{2cr}\right) & \\ \quad -\frac{1}{2}\sqrt{(-c+r+R)(c+R-r)(c+r-R)(c+r+R)} & \text{for c} < (r+R) \end{cases}$$

$$q = z \cdot C_D \cdot A \sqrt{\frac{2}{\rho}} \cdot \frac{p}{\left(p^2 + p_{cr}^2\right)^{1/4}}$$

$$\Delta p = p_A - p_B,$$

where

q	Flow rate
p	Pressure differential
p_A, p_B	Gauge pressures at the block terminals
A	Instantaneous passage area between orifices
c	Instantaneous distance between hole centers
r	Smaller hole radius
R	Larger hole radius
z	Number of hole pairs
C_D	Flow discharge coefficient
ρ	Fluid density
A_{leak}	Closed orifice leakage area
p_{cr}	Minimum pressure for turbulent flow

The minimum pressure for turbulent flow, p_{cr}, is calculated according to the laminar transition specification method:

- By pressure ratio — The transition from laminar to turbulent regime is defined by the following equations:

  $$p_{cr} = (p_{avg} + p_{atm})(1 - B_{lam})$$

  $$p_{avg} = (p_A + p_B)/2$$

  where

p_{avg}	Average pressure between the block terminals
p_{atm}	Atmospheric pressure, 101325 Pa

B_{lam}	Pressure ratio at the transition between laminar and turbulent regimes (**Laminar flow pressure ratio** parameter value)

- By Reynolds number — The transition from laminar to turbulent regime is defined by the following equations:

$$p_{cr} = \frac{\pi}{4A}\left(\frac{\text{Re}_{cr}}{C_D}\right)^2 \cdot \rho \cdot \frac{\nu^2}{2}$$

where

ν	Fluid kinematic viscosity
Re_{cr}	Critical Reynolds number (**Critical Reynolds number** parameter value)

The block positive direction is from port A to port B. This means that the flow rate is positive if it flows from A to B. Positive signal at port S moves the sleeve in the positive direction.

Basic Assumptions and Limitations

Inertial effects are not taken into account.

Parameters

Sleeve hole diameter

Diameter of the holes drilled in the sleeve. The default value is `0.005` m.

Case hole diameter

Diameter of the holes drilled in the case. The default value is `0.0054` m.

Number of interacting pairs

Number of interacting hole pairs. The default value is `1`.

Flow discharge coefficient

Semi-empirical parameter for orifice capacity characterization. Its value depends on the geometrical properties of the orifice, and usually is provided in textbooks or manufacturer data sheets. The default value is `0.7`.

Laminar transition specification

Select how the block transitions between the laminar and turbulent regimes:

- `Pressure ratio` — The transition from laminar to turbulent regime is smooth and depends on the value of the **Laminar flow pressure ratio** parameter. This method provides better simulation robustness.
- `Reynolds number` — The transition from laminar to turbulent regime is assumed to take place when the Reynolds number reaches the value specified by the **Critical Reynolds number** parameter.

Laminar flow pressure ratio

Pressure ratio at which the flow transitions between laminar and turbulent regimes. The default value is `0.999`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Pressure ratio`.

Critical Reynolds number

The maximum Reynolds number for laminar flow. The value of the parameter depends on the orifice geometrical profile. You can find recommendations on the parameter value in hydraulics textbooks. The default value is `12`. This parameter is visible only if the **Laminar transition specification** parameter is set to `Reynolds number`.

Leakage area

The total area of possible leaks in the completely closed orifice. The main purpose of the parameter is to maintain numerical integrity of the circuit by preventing a portion of the system from getting isolated after the orifice is completely closed. The parameter value must be greater than 0. The default value is `1e-12 m^2`.

Global Parameters

Parameters determined by the type of working fluid:

- **Fluid density**
- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

A

Hydraulic conserving port associated with the orifice inlet.

B

Hydraulic conserving port associated with the orifice outlet.

S

Physical signal port that provides the instantaneous value of the distance between the hole centers.

See Also

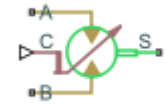
Annular Orifice | Orifice with Variable Area Round Holes | Orifice with Variable Area Slot | Variable Area Hydraulic Orifice | Variable Orifice

Introduced in R2014b

Variable-Displacement Motor

Variable-displacement bidirectional hydraulic motor

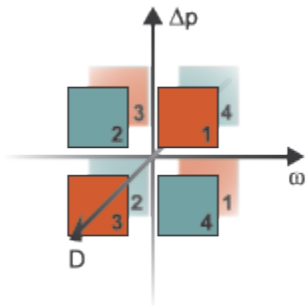
Library: Simscape / Fluids / Hydraulics (Isothermal) / Pumps and Motors



Description

The Variable-Displacement Motor block represents a device that extracts power from a hydraulic (isothermal liquid) network and delivers it to a mechanical rotational network. The motor displacement varies in proportion to the physical signal input specified at port C or D. The exact port used depends on the block variant selected. See “Ports” on page 1-1313.

Ports **A** and **B** represent the motor inlet and outlet, respectively. Port **S** represents the motor drive shaft. During normal operation, the angular velocity at port **S** is positive if the pressure drop from port **A** to port **B** is positive also. This operation mode is referred to here as forward motor.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure drop from port **A** to port **B** (Δp), on the angular velocity at port **S** (ω), and on the

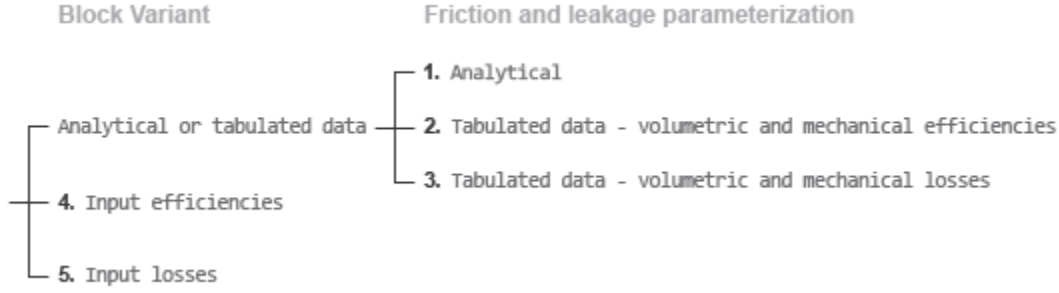
instantaneous volumetric displacement of the component (*D*). The “Operation Modes” on page 1-1304 figure maps the modes to the octants of a Δ*p*-*ω*-*D* chart. The modes are labeled 1–4:

- Mode **1**: forward motor — A positive pressure drop generates a positive shaft angular velocity.
- Mode **2**: reverse pump — A negative shaft angular velocity generates a negative pressure gain (shown in the figure as a positive pressure drop).
- Mode **3**: reverse motor — A negative pressure drop generates a negative shaft angular velocity.
- Mode **4**: forward pump — A positive shaft angular velocity generates a positive pressure gain (shown in the figure as a negative pressure drop).

The response time of the motor is considered negligible in comparison with the system response time. The motor is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The motor model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the motor inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Displacement Parameterizations

The displacement volume input depends on the block variant selected. If the active block variant is `Input efficiencies` or `Input losses`, the block obtains the instantaneous displacement volume directly from the physical signal input at port D.

If the active block variant is `Analytical or tabulated data`, the block computes the instantaneous displacement volume from the control member position specified at port C. This computation depends on the **Displacement parameterization** parameter setting:

- `Maximum displacement and control member stroke` — Compute the displacement volume per unit rotation as a linear function of the control member position specified at port C.
- `Displacement vs. control member position table` — Compute the displacement volume per unit volume using interpolation or extrapolation of displacement tabular data specified at discrete control member positions.

Flow Rate and Driving Torque

The volumetric flow rate generated at the motor is

$$q = q_{\text{Ideal}} + q_{\text{Leak}},$$

where:

- q is the net volumetric flow rate.
- q_{Ideal} is the ideal volumetric flow rate.
- q_{Leak} is the internal leakage volumetric flow rate.

The torque generated at the motor is

$$\tau = \tau_{\text{Ideal}} - \tau_{\text{Friction}},$$

where:

- τ is the net torque.
- τ_{Ideal} is the ideal torque.
- τ_{Friction} is the friction torque.

The ideal volumetric flow rate is

$$q_{\text{Ideal}} = D_{\text{Sat}} \cdot \omega,$$

and the ideal generated torque is

$$\tau_{\text{Ideal}} = D_{\text{Sat}} \cdot \Delta p,$$

where:

- D_{Sat} is a smoothed displacement computed so as to remove numerical discontinuities between negative and positive displacements.
- ω is the instantaneous angular velocity of the rotary shaft.
- Δp is the instantaneous pressure drop from inlet to outlet.

The saturation displacement depends on the block variant selected. If the active variant is `Analytical or tabulated data`,

$$D_{\text{Sat}} = \begin{cases} \text{sign}(D) \cdot D_{\text{Max}}, & |D| \geq D_{\text{Max}} \\ \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases},$$

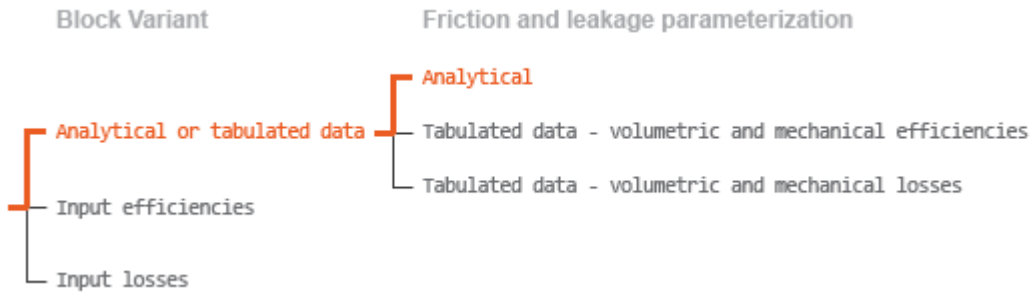
where:

- D is the instantaneous fluid displacement determined from the physical signal input specified at port **C** or port **D**.
- D_{Max} is the specified value of the **Maximum displacement** block parameter.
- $D_{\text{Threshold}}$ is the specified value of the **Displacement threshold for motor-pump transition** block parameter.

If the active variant is `Input efficiencies` or `Input losses`, there is no upper bound on the displacement input and the saturation displacement reduces to:

$$D_{\text{Sat}} = \begin{cases} \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases}.$$

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$q_{\text{Leak}} = K_{\text{HP}} \Delta p,$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}} \left| \frac{D_{\text{Sat}}}{D_{\text{Max}}} \right| |\Delta p| \right) \tanh\left(\frac{4\omega}{\omega_{\text{Thresh}}} \right),$$

where:

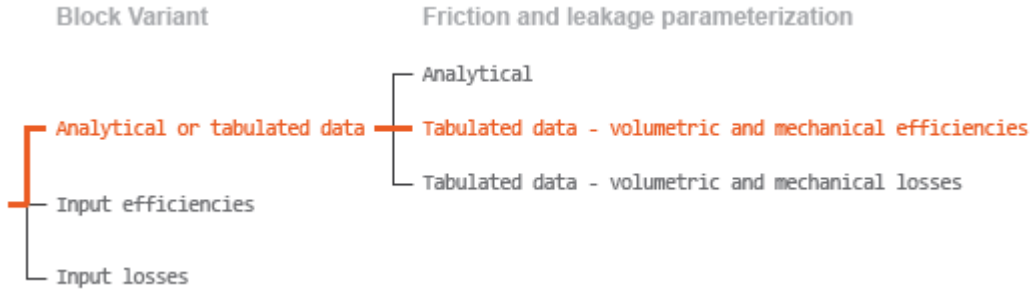
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- $\omega_{\text{Threshold}}$ is the threshold angular velocity for the motor-pump transition. The threshold angular velocity is an internally set fraction of the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{\text{HP}} = \frac{\nu_{\text{Nom}}}{\rho \upsilon} \frac{\rho_{\text{Nom}} \omega_{\text{Nom}} D_{\text{Max}}}{\Delta p_{\text{Nom}}} \left(\frac{1}{\eta_{\text{v,Nom}}} - 1 \right),$$

where:

- ν_{Nom} is the **Nominal kinematic viscosity** parameter. This is the kinematic viscosity at which the nominal volumetric efficiency is specified.
- ρ_{Nom} is the **Nominal fluid density** parameter. This is the density at which the nominal volumetric efficiency is specified.
- ρ is the actual fluid density in the attached hydraulic (isothermal liquid) network. This density can differ from the **Nominal fluid density** parameter.
- ω_{Nom} is the **Nominal shaft angular velocity** parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- ν is the fluid kinematic viscosity in the attached hydraulic fluid network.
- Δp_{Nom} is the **Nominal pressure drop** parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{\text{v,Nom}}$ is the **Volumetric efficiency at nominal conditions** parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`, the leakage flow rate is

$$q_{\text{Leak}} = q_{\text{Leak,Motor}} \frac{(1+\alpha)}{2} + q_{\text{Leak,Pump}} \frac{(1-\alpha)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Motor}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Pump}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the motor-pump transition.
- $q_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $q_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure drop threshold for motor-pump transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.
- $D_{\text{Threshold}}$ is the specified value of the **Displacement threshold for motor-pump transition** block parameter.

The leakage flow rate is computed from efficiency tabulated data through the equation

$$q_{\text{Leak,Motor}} = (1-\eta_{\text{v}})\,q,$$

in motor mode and through the equation

$$q_{\text{Leak,Pump}} = -(1-\eta_{\text{v}})\,q_{\text{Ideal}}.$$

in pump mode, where:

- η_{v} is the volumetric efficiency obtained through interpolation or extrapolation of the **Volumetric efficiency table, e_v(dp,w,D)** parameter data.

Similarly, the friction torque is computed from efficiency tabulated data through the equation

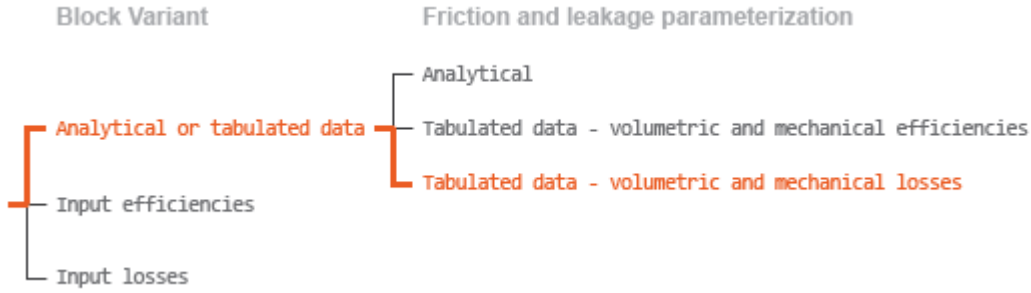
$$\tau_{\text{Friction,Motor}} = (1-\eta_{\text{m}})\,\tau_{\text{Ideal}},$$

in motor mode and through the equation

$$\tau_{\text{Friction,Pump}} = -(1-\eta_{\text{m}})\,\tau.$$

in pump mode, where:

- η_{m} is the mechanical efficiency obtained through interpolation or extrapolation of the **Mechanical efficiency table, e_m(dp,w,D)** parameter data.



Case 3: Loss Tabulated Data

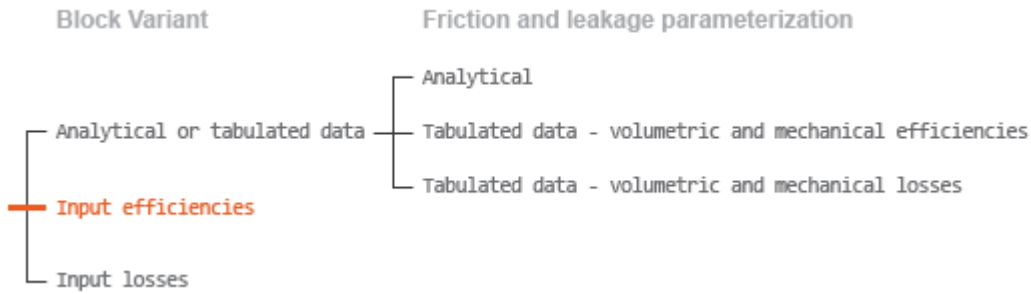
If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical losses`, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}\left(\Delta p, \omega, D_{\text{Sat}}\right).$$

and the friction torque equation is

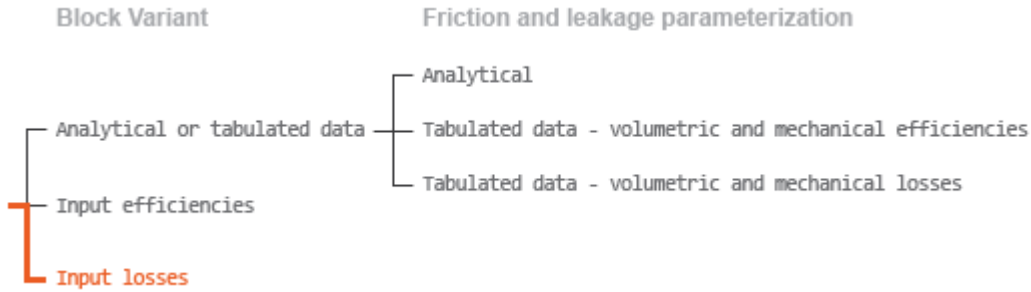
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega, D_{\text{Sat}}\right),$$

where $q_{\text{Leak}}(\Delta p,\omega,D_{\text{Sat}})$ and $\tau_{\text{Friction}}(\Delta p,\omega,D_{\text{Sat}})$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the **Volumetric loss table, q_loss(dp,w)** and **Mechanical loss table, torque_loss (dp,w)** parameter data.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

Assumptions

- Fluid compressibility is negligible.
- Loading on the motor shaft due to inertia, friction, and spring forces is negligible.

Ports

Input

`C` — Control member position, `m`

physical signal

Physical signal input port for the control member position. The block maps the control member position to the corresponding displacement volume using the tabulated data specified in the block dialog box.

Dependencies

This port is exposed only when the block variant is set to `Analytical or tabulated data`.

`D` — Displacement volume, `cm^3/rev`

physical signal

Physical signal input port for the volume of fluid displaced per unit rotation. A smoothing function eases the transition between positive and negative input values.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies` or `Input losses`.

`EV` — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

`EM` — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

`LV` — Volumetric loss, `m^3/s`

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the motor inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

`LM` — Mechanical loss, `N*m`

physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotating motor shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

`A` — Motor inlet

hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the motor inlet.

`B` — Motor outlet

hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the motor outlet.

`S` — Motor shaft

mechanical rotational

Mechanical rotational conserving port representing the motor shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-1305.

Variant 1: Analytical or tabulated data

`Displacement parameterization` — Parameterization used to map control positions to displacements

`Maximum displacement and control member stroke` (default)

Parameterization used to convert the physical signal at input port C to an instantaneous displacement:

- `Maximum displacement and control member stroke` — Compute the displacement as a linear function of the control position specified at port C.
- `Displacement vs. control member position table` — Obtain the displacement through interpolation or extrapolation of tabulated data specified over a range of control positions.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data`.

Restricted Parameter

This parameter is locked for editing when using the Simscape Restricted mode.

`Maximum stroke` — Control position corresponding to maximum displacement

`0.005 m` (default) | scalar with units of length

Control position corresponding to a maximum displacement. The physical signal input at port C saturates at this value. If the input rises higher, the block sets the control position to the maximum stroke.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Maximum displacement and control member stroke`.

`Maximum displacement` — Maximum fluid volume swept per unit shaft rotation

`30 cm^3/rev` (default) | scalar with units of volume/angle

Maximum fluid volume swept per unit shaft rotation. The displacement reaches this value when the control position hits the maximum stroke.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Maximum displacement and control member stroke`.

`Control member position vector` — Control positions at which to specify displacement tabular data

`[-0.0075, -0.0025, 0.0, 0.0025, 0.0075] m` (default) | *M*-element vector with units of length

M-element vector of control positions at which to specify the instantaneous displacement. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

`Displacement vector` — Displacement tabular data for the specified control positions

`[-30.0, -19.0, 0.0, 19.0, 30.0] cm^3/rev` (default) | *M*-element vector with units of volume/angle

M-element vector with the displacement tabular data for the specified control positions. The vector size, *M*, must match that of the **Control member position vector** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

`Interpolation method` — Numerical technique used to determine displacements within the tabular data range

`Linear` (default) | `Smooth`

Numerical technique used to map control position signals to displacements within the tabular date range. The interpolation method joins tabular data points using straight or curved line segments. Displacements within the range of the tabular data are assumed to lie on these segments.

- `Linear` — Join neighboring data points using straight line segments. Line slopes are generally discontinuous at the line segment end points.
- `Smooth` — Join neighboring data points using curved line segments generated with a modified Akima algorithm. Line slopes are continuous at the line segment end points.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

Restricted Parameter

This parameter is locked for editing when using the Simscape Restricted mode.

`Extrapolation method` — Numerical technique used to determine displacements outside the tabular data range

`Linear` (default) | `Nearest`

Numerical technique used to map control position signals to displacements outside the tabular date range. The extrapolation method extends the first and last tabular data points outward using horizontal or sloped straight line segments. Displacements outside the range of the tabular data are assumed to lie on these segments.

- `Linear` — Extend the tabular data using sloped straight line segments. The line slopes are computed from the first and last two tabular data points.
- `Nearest` — Extend the tabular data using horizontal straight line segments. The lines correspond to the displacements specified in the first and last tabular data points.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

Restricted Parameter

This parameter is locked for editing when using the Simscape Restricted mode.

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque

`Analytical` (default)

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup tables to map pressure drop, angular velocity, and displacement to component efficiencies or losses. The tabular options include:

- `Tabulated data – volumetric and mechanical efficiencies`
- `Tabulated data – volumetric and mechanical losses`

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data`.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the volumetric efficiency

`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotating shaft at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure drop` — Pressure drop at which to specify the volumetric efficiency

`100e5 Pa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal kinematic viscosity` — Kinematic viscosity at which to specify the volumetric efficiency

`18 cSt` (default) | scalar with units of area/time

Kinematic viscosity of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating

conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal fluid density` — Fluid density at which to specify the volumetric efficiency

`900 kg/m^3` (default) | scalar with units of mass/volume

Mass density of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions

`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation

`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure drop coefficient at maximum displacement` — Proportionality constant at maximum displacement between friction torque and pressure drop
`0.6e-6 N*m/Pa` (default) | scalar with units of torque/pressure

Proportionality constant at maximum displacement between the friction torque on the mechanical shaft and the pressure drop from inlet to outlet.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Displacement threshold for motor-pump transition` — Displacement below which the transition between motoring and pumping modes begins
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

`Pressure drop vector for efficiencies, dp` — Pressure drops at which to specify the volumetric and mechanical efficiencies

M-element vector with units of pressure (default)

M-element vector of pressure drops at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies

N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Displacement vector for efficiencies, D` — Displacements at which to specify the volumetric and mechanical efficiencies

L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

`Volumetric efficiency table, e_v(dp,w,D)` — Volumetric efficiencies at the specified pressure drops, angular velocities, and displacements
unitless *M*-by-*N*-by*L* matrix

M-by-*N*-by-*L* matrix with the volumetric efficiencies at the specified fluid pressure drops, shaft angular velocities, and displacements. The efficiencies must fall in the range of 0–1. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w,D)` — Mechanical efficiencies at the specified pressure drops, angular velocities, and displacements
unitless *M*-by-*N* matrix

M-by-*N*-by-*L* matrix with the mechanical efficiencies corresponding to the specified fluid pressure drops, shaft angular velocities, and displacements. The efficiencies must fall in the range of 0–1. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes
`1e5 Pa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Angular velocity threshold for motor-pump transition` — Shaft angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Check if operating beyond the octants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure drop, shaft angular velocity, or instantaneous displacement cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated`

`data — volumetric and mechanical efficiencies` or `Tabulated data — volumetric and mechanical losses`.

`Pressure drop vector for losses, dp` — Pressure drops at which to specify the volumetric and mechanical losses
M-element vector with units of pressure (default)

M-element vector of pressure drops at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses
N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Displacement vector for losses, D` — Displacements at which to specify the volumetric and mechanical losses
L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

`Volumetric loss table, q_loss(dp,w,D)` — Internal leakage flow rates at the specified pressure drops, angular velocities, and displacements
M-by-*N*-by-*L* matrix with units of volume/time

M-by-*N*-by-*L* matrix with the volumetric losses at the specified fluid pressure drops, shaft angular velocities, and displacements. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Mechanical loss table, torque_loss(dp,w,D)` — Friction torques at the specified pressure drops, angular velocities, and displacements

M-by-*N*-by-*L* matrix with units of torque

M-by-*N*-by-*L* matrix with the mechanical losses at the specified fluid pressure drops, shaft angular velocities, and displacements. Mechanical loss is defined here as the friction torque due to seals and internal components. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes

`1e5 Pa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for motor-pump transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes

`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Displacement threshold for motor-pump transition` — Displacement at which to initiate a smooth transition between pump and motor modes

`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at this value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure

`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

Variant 3: Input losses

`Displacement threshold for motor-pump transition` — Displacement below which the transition between motoring and pumping modes begins
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Check if operating beyond the motor mode` — Simulation warning mode for operating conditions outside motoring mode
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the motoring mode. A warning is issued if the motor transitions to pumping mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Input losses`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be

useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

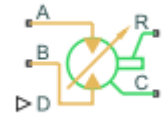
This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

See Also

Fixed-Displacement Motor | Fixed-Displacement Motor (TL) | Fixed-Displacement Pump | Fixed-Displacement Pump (TL) | Variable-Displacement Pump

Variable-Displacement Motor (TL)

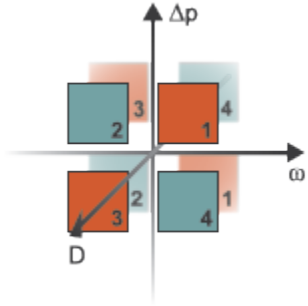
Variable-displacement bidirectional thermal liquid motor
Library: Simscape / Fluids / Thermal Liquid / Pumps & Motors



Description

The Variable-Displacement Motor block represents a device that extracts power from a thermal liquid network and delivers it to a mechanical rotational network. The motor displacement varies during simulation according to the physical signal input specified at port **D**.

Ports **A** and **B** represent the motor inlets. Ports **R** and **C** represent the motor drive shaft and case. During normal operation, a pressure drop from port **A** to port **B** causes a positive flow rate from port **A** to port **B** and a positive rotation of the motor shaft relative to the motor case. This operation mode is referred to here as *forward motor*.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure drop from port **A** to port **B** (Δp), on the angular velocity at port **R** relative to port **C** (ω),

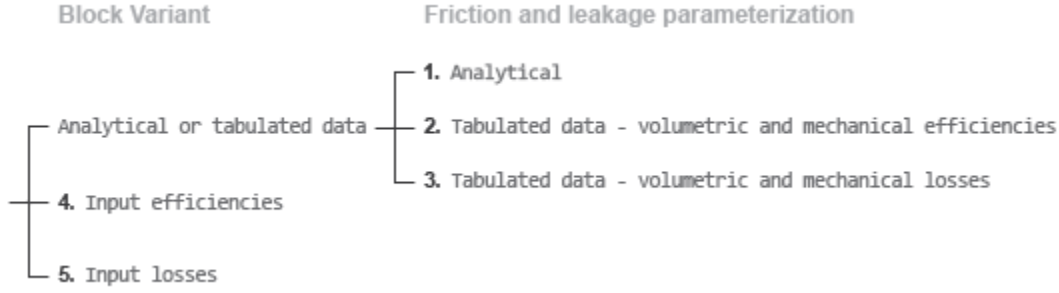
and on the instantaneous displacement of the component (*D*). The “Operation Modes” on page 1-1331 figure maps the modes to the octants of a Δ*p*-*ω*-*D* chart. The modes are labeled 1–4:

- Mode **1**: forward motor — A positive pressure drop generates a positive shaft angular velocity.
- Mode **2**: reverse pump — A negative shaft angular velocity generates a negative pressure gain (shown in the figure as a positive pressure drop).
- Mode **3**: reverse motor — A negative pressure drop generates a negative shaft angular velocity.
- Mode **4**: forward pump — A positive shaft angular velocity generates a positive pressure gain (shown in the figure as a negative pressure drop).

The response time of the motor is considered negligible in comparison with the system response time. The motor is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The motor model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the motor inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Flow Rate and Driving Torque

The mass flow rate generated at the motor is

$$\dot{m} = \dot{m}_{\mathrm{Ideal}} + \dot{m}_{\mathrm{Leak}},$$

where:

- $\dot{m}$ is the actual mass flow rate.
- $\dot{m}_{\mathrm{Ideal}}$ is the ideal mass flow rate.
- $\dot{m}_{\mathrm{Leak}}$ is the internal leakage mas flow rate.

The torque generated at the motor is

$$\tau = \tau_{\mathrm{Ideal}} - \tau_{\mathrm{Friction}},$$

where:

- τ is the actual torque.
- τ_{Ideal} is the ideal torque.
- τ_{Friction} is the friction torque.

The ideal mass flow rate is

$$\dot{m}_{\text{Ideal}} = \rho D_{\text{Sat}} \omega,$$

and the ideal generated torque is

$$\tau_{\text{Ideal}} = D_{\text{Sat}} \Delta p,$$

where:

- ρ is the average of the fluid densities at thermal liquid ports **A** and **B**.
- D_{Sat} is a smoothed displacement computed so as to remove numerical discontinuities between negative and positive displacements.
- ω is the shaft angular velocity.
- Δp is the pressure drop from inlet to outlet.

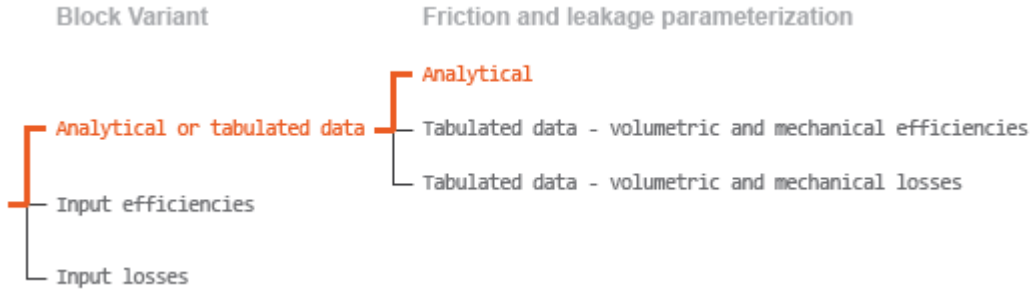
The saturation displacement is defined as:

$$D_{\text{Sat}} = \begin{cases} \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases}.$$

where:

- D is the displacement specified at physical signal port **D**.
- $D_{\text{Threshold}}$ is the specified value of the **Displacement threshold for motor-pump transition** block parameter.

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$\dot{m}_{\mathrm{Leak}} = \frac{K_{\mathrm{HP}}\rho_{\mathrm{Avg}}\Delta p}{\mu_{\mathrm{Avg}}},$$

and the friction torque is

$$\tau_{\mathrm{Friction}} = \left(\tau_0 + K_{\mathrm{TP}}\left|\Delta p\right|\frac{\left|D_{\mathrm{Sat}}\right|}{D_{\mathrm{Nom}}}\tanh\frac{4\omega}{(5e-5)\omega_{\mathrm{Nom}}}\right),$$

where:

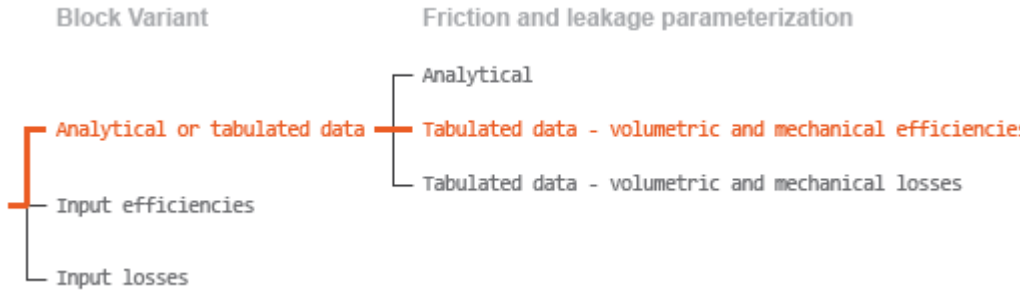
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- μ is the dynamic viscosity of the thermal liquid, taken here as the average of its values at the thermal liquid ports.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- D_{Nom} is the specified value of the **Nominal Displacement** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{HP} = \frac{D_{Sat}\omega_{Nom}\mu_{Nom}\left(\frac{1}{\eta_{v,Nom}} - 1\right)}{\Delta p_{Nom}},$$

where:

- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- μ_{Nom} is the specified value of the **Nominal Dynamic viscosity** block parameter. This is the dynamic viscosity at which the nominal volumetric efficiency is specified.
- Δp_{Nom} is the specified value of the **Nominal pressure drop** block parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{v,Nom}$ is the specified value of the **Volumetric efficiency at nominal conditions** block parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`, the leakage flow rate is

$$\dot{m}_{Leak} = \dot{m}_{Leak,Motor}\frac{(1+\alpha)}{2} + \dot{m}_{Leak,Pump}\frac{(1-\alpha)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Motor}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Pump}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the motor-pump transition.
- $\dot{m}_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $\dot{m}_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure drop threshold for motor-pump transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.
- $D_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.

The leakage flow rate is calculated from the volumetric efficiency, a quantity that is specified in tabulated form over the Δp–ω–D domain via the **Volumetric efficiency table** block parameter. When operating in motor mode (quadrants **1** and **3** of the Δp–ω–D chart shown in the “Operation Modes” on page 1-542 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak,Motor}} = \left(1-\eta_{\text{v}}\right)\dot{m},$$

where η_{v} is the volumetric efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in pump mode (quadrants **2** and **4** of the Δp–ω–D chart), the leakage flow rate is:

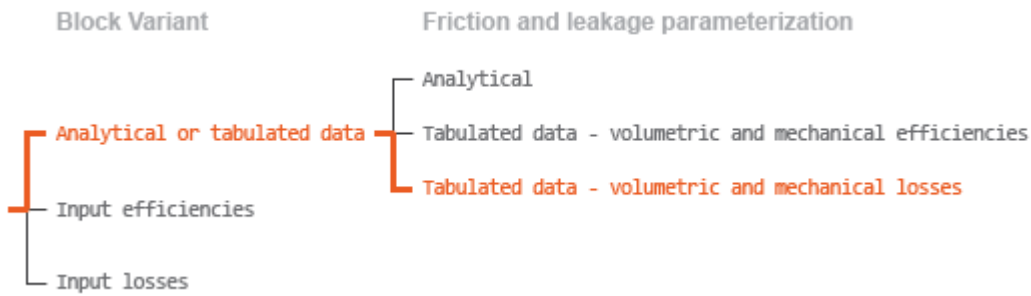
$$\dot{m}_{\text{Leak,Pump}} = -(1-\eta_{\text{v}})\dot{m}_{\text{Ideal}}.$$

The friction torque is similarly calculated from the mechanical efficiency, a quantity that is specified in tabulated form over the Δp–ω–D domain via the **Mechanical efficiency table** block parameter. When operating in motor mode (quadrants **1** and **3** of the Δp–ω–D chart):

$$\tau_{\text{Friction,Motor}} = (1-\eta_{\text{m}})\tau_{\text{Ideal}},$$

where η_{m} is the mechanical efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in pump mode (quadrants **2** and **4** of the Δp–ω–D chart):

$$\tau_{\text{Friction,Pump}} = -(1-\eta_{\text{m}})\tau.$$



Case 3: Loss Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`, the leakage (volumetric) flow rate is specified directly in tabulated form over the Δp–ω–D domain:

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega, D_{\text{Sat}}).$$

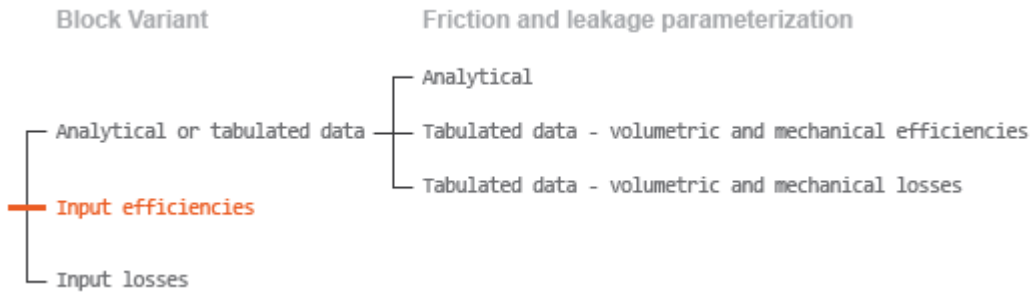
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega, D_{\text{Sat}}\right),$$

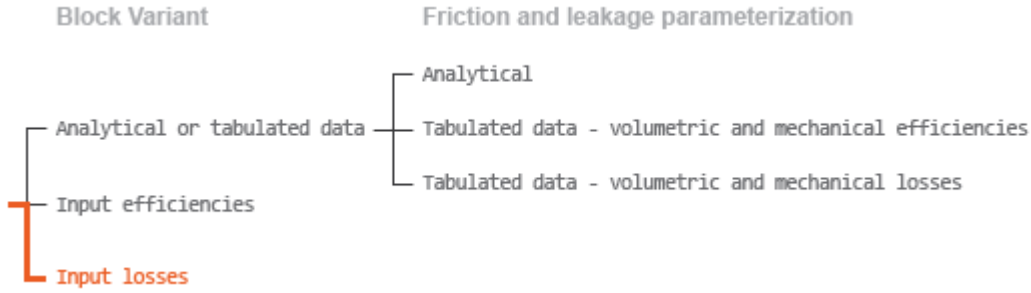
where $q_{\text{Leak}}(\Delta p,\omega)$ and $\tau_{\text{Friction}}(\Delta p,\omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.

The efficiencies are defined as positive quantities with value between zero and one. Input values outside of these bounds are set equal to the nearest bound (zero for inputs smaller than zero, one for inputs greater than one). In other words, the efficiency signals are *saturated* at zero and one.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

The signs of the inputs are ignored. The block sets the signs automatically from the operating conditions established during simulation—more precisely, from the Δp–ω quadrant in which the component happens to be operating. In other words, whether an input is positive or negative is irrelevant to the block.

Assumptions

- Fluid compressibility is negligible.
- Loading on the motor shaft due to inertia, friction, and spring forces is negligible.

Ports

Input

`D` — Displacement volume, `cm^3/rev`
physical signal

Physical signal input port for the volume of fluid displaced per unit rotation. A smoothing function eases the transition between positive and negative input values.

EV — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

EM — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

LV — Volumetric loss, `m^3/s`

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the motor inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

LM — Mechanical loss, `N*m`

physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotating motor shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

A — Motor inlet
thermal liquid

Thermal liquid conserving port representing the motor inlet.

B — Motor outlet
thermal liquid

Thermal liquid conserving port representing the motor outlet.

C — Motor Case
mechanical rotational

Mechanical rotational conserving port representing the motor case.

R — Motor Shaft
mechanical rotational

Mechanical rotational conserving port representing the rotational motor shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-1305.

Variant 1: Analytical or tabulated data

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque
`Analytical` (default) | `Tabulated data — volumetric and mechanical efficiencies` | `Tabulated data — volumetric and mechanical losses`

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup tables to map pressure drop, angular velocity, and displacement to component efficiencies or losses.

`Nominal displacement` — Displacement at which to specify the volumetric efficiency

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid displacement at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the volumetric efficiency

`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotary shaft at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure drop` — Pressure drop at which to specify the volumetric efficiency

`10 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal dynamic viscosity` — Dynamic viscosity at which to specify the volumetric efficiency
`0.9 cP` (default) | scalar with units of area/time

Dynamic viscosity of the hydraulic fluid at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions
`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation
`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure drop coefficient at nominal displacement` — Proportionality constant at maximum displacement between friction torque and pressure drop
`0.6 N*m/MPa` (default) | scalar with units of torque/pressure

Proportionality constant at maximum displacement between the friction torque on the mechanical shaft and the pressure drop from inlet to outlet.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Displacement threshold for motor-pump transition` — Fluid displacement at which to initiate a smooth transition between pump and motor modes

`0.5 cm^3/rev` (default) | positive scalar with units of volume/area

Fluid displacement below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet

`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Pressure drop vector for efficiencies, dp` — Pressure drops at which to specify the volumetric and mechanical efficiencies

M-element vector with units of pressure (default)

M-element vector of pressure drops at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies

N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Displacement vector for efficiencies, D` — Displacements at which to specify the volumetric and mechanical efficiencies

L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Volumetric efficiency table, e_v(dp,w,D)` — Volumetric efficiencies at the specified pressure drops, angular velocities, and displacements

unitless *M*-by-*N*-by*L* matrix

M-by-*N*-by-*L* matrix with the volumetric efficiencies at the specified fluid pressure drops, shaft angular velocities, and displacements. The efficiencies must be in the range of `0`–`1`. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w,D)` — Mechanical efficiencies at the specified pressure drops, angular velocities, and displacements

unitless *M*-by-*N* matrix

M-by-*N*-by-*L* matrix with the mechanical efficiencies corresponding to the specified fluid pressure drops, shaft angular velocities, and displacements. The efficiencies must be in the range of `0`–`1`. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes

`1e-3 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Angular velocity threshold for motor-pump transition` — Shaft angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Displacement threshold for motor-pump transition` — Fluid displacement at which to initiate a smooth transition between pump and motor modes
`0.5 cm^3/rev` (default) | positive scalar with units of volume/area

Fluid displacement below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Check if operating beyond the octants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure drop, shaft angular velocity, or instantaneous displacement cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies` or `Tabulated data – volumetric and mechanical losses`.

`Pressure drop vector for losses, dp` — Pressure drops at which to specify the volumetric and mechanical losses

M-element vector with units of pressure (default)

M-element vector of pressure drops at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses

N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical losses`.

`Displacement vector for losses, D` — Displacements at which to specify the volumetric and mechanical losses

L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

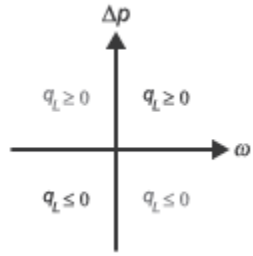
`Volumetric loss table, q_loss(dp,w,D)` — Internal leakage flow rates at the specified pressure drops, angular velocities, and displacements

M-by-*N*-by-*L* matrix with units of volume/time

M-by-*N*-by-*L* matrix with the volumetric losses at the specified fluid pressure drops, shaft angular velocities, and displacements. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants. The tabulated data for the volumetric losses must obey the convention shown in the figure, with positive values at positive pressure drops and negative values at negative pressure drops.



Dependencies

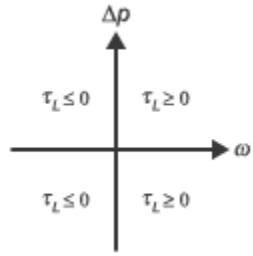
This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Mechanical loss table, torque_loss(dp,w,D)` — Friction torques at the specified pressure drops, angular velocities, and displacements
M-by-*N*-by-*L* matrix with units of torque

M-by-*N*-by-*L* matrix with the mechanical losses at the specified fluid pressure drops, shaft angular velocities, and displacements. Mechanical loss is defined here as the friction torque due to seals and internal components. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure drop vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants. The tabulated data for the mechanical losses must obey the convention shown in the figure, with positive values at positive angular velocities and negative values at negative angular velocities.



Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal
`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal
`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at this value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for motor-pump transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Displacement threshold for motor-pump transition` — Displacement at which to initiate a smooth transition between pump and motor modes
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

Variant 3: Input losses

`Pressure drop threshold for motor-pump transition` — Pressure drop at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure drop from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Angular velocity threshold for motor-pump transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Displacement threshold for motor-pump transition` — Displacement at which to initiate a smooth transition between pump and motor modes
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Check if operating beyond the motor mode` — Simulation warning mode for operating conditions outside motoring mode
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the motoring mode. A warning is issued if the motor transitions to pumping mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Variables

`Mass flow rate into port A` — Mass flow rate into the component through thermal liquid port A
`1 kg/s` (default) | scalar with units of mass/time

Mass of fluid entering the component through the inlet per unit time at the start of simulation.

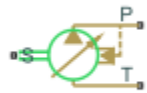
See Also

Fixed-Displacement Motor (TL) | Fixed-Displacement Pump (TL)

Introduced in R2017b

Variable-Displacement Pressure-Compensated Pump

Hydraulic pump maintaining preset pressure at outlet by regulating its flow delivery



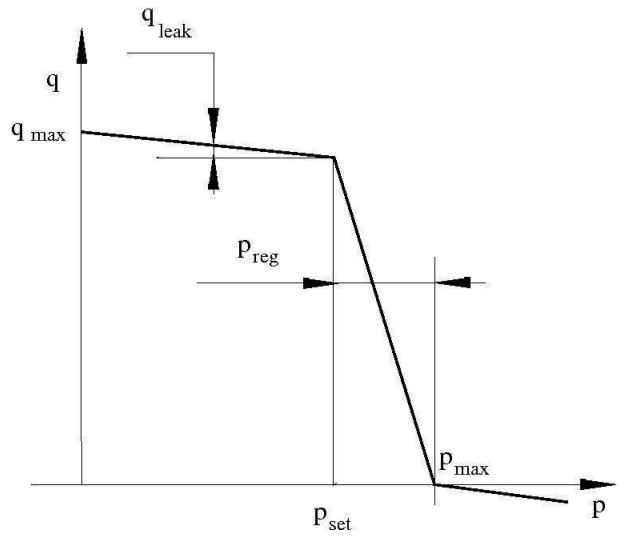
Library

Pumps and Motors

Description

The Variable-Displacement Pressure-Compensated Pump block represents a positive, variable-displacement, pressure-compensated pump of any type as a data-sheet-based model. The key parameters required to parameterize the block are the pump maximum displacement, regulation range, volumetric and total efficiencies, nominal pressure, and angular velocity. All these parameters are generally provided in the data sheets or catalogs.

The following figure shows the delivery-pressure characteristic of the pump.



The pump tries to maintain preset pressure at its outlet by adjusting its delivery flow in accordance with the system requirements. If pressure differential across the pump is less than the setting pressure, the pump outputs its maximum delivery corrected for internal leakage. After the pressure setting has been reached, the output flow is regulated to maintain preset pressure by changing the pump's displacement. The displacement can be changed from its maximum value down to zero, depending upon system flow requirements. The pressure range between the preset pressure and the maximum pressure, at which the displacement is zero, is referred to as regulation range. The smaller the range, the higher the accuracy at which preset pressure is maintained. The range size also affects the pump stability, and decreasing the range generally causes stability to decrease.

The variable-displacement, pressure-compensated pump is represented with the following equations:

$$q = D \cdot \omega - k_{leak} \cdot p$$

$$T = D \cdot p / \eta_{mech}$$

$$D = \begin{cases} D_{\max} & \text{for } p <= p_{set} \\ D_{\max} - K(p - p_{set}) & \text{for } p_{set} < p < p_{\max} \\ 0 & \text{for } p >= p_{\max} \end{cases}$$

$$p_{\max} = p_{set} + p_{reg}$$

$$K = D_{\max} / (p_{\max} - p_{set})$$

$$k_{leak} = \frac{k_{HP}}{\nu \cdot \rho}$$

$$k_{HP} = \frac{D_{\max} \cdot \omega_{nom} (1 - \eta_V) \cdot \nu_{nom} \cdot \rho_{nom}}{p_{nom}}$$

$$p = p_P - p_T$$

where

q	Pump delivery
p	Pressure differential across the pump
p_P, p_T	Gauge pressures at the block terminals
D	Pump instantaneous displacement
D_{max}	Pump maximum displacement
p_{set}	Pump setting pressure
p_{max}	Maximum pressure, at which the pump displacement is zero
T	Torque at the pump driving shaft
ω	Pump angular velocity
k_{leak}	Leakage coefficient
k_{HP}	Hagen-Poiseuille coefficient
η_V	Pump volumetric efficiency
η_{mech}	Pump mechanical efficiency

ν	Fluid kinematic viscosity
ρ	Fluid density
ρ_{nom}	Nominal fluid density
p_{nom}	Pump nominal pressure
ω_{nom}	Pump nominal angular velocity
ν_{nom}	Nominal fluid kinematic viscosity

The leakage flow is determined based on the assumption that it is linearly proportional to the pressure differential across the pump and can be computed by using the Hagen-Poiseuille formula

$$p = \frac{128\mu l}{\pi d^4} q_{leak} = \frac{\mu}{k_{HP}} q_{leak}$$

where

q_{leak}	Leakage flow
$d,\ l$	Geometric parameters of the leakage path
μ	Fluid dynamic viscosity, $\mu = \nu \cdot \rho$

The leakage flow at $p = p_{nom}$ and $\nu = \nu_{nom}$ can be determined from the catalog data

$$q_{leak} = D_{max} \cdot \omega_{nom} \left(1 - \eta_V\right)$$

which provides the formula to determine the Hagen-Poiseuille coefficient

$$k_{HP} = \frac{D_{max} \cdot \omega_{nom} \left(1 - \eta_V\right) \cdot \nu_{nom} \cdot \rho_{nom}}{p_{nom}}$$

The pump mechanical efficiency is not usually available in data sheets, therefore it is determined from the total and volumetric efficiencies by assuming that the hydraulic efficiency is negligibly small

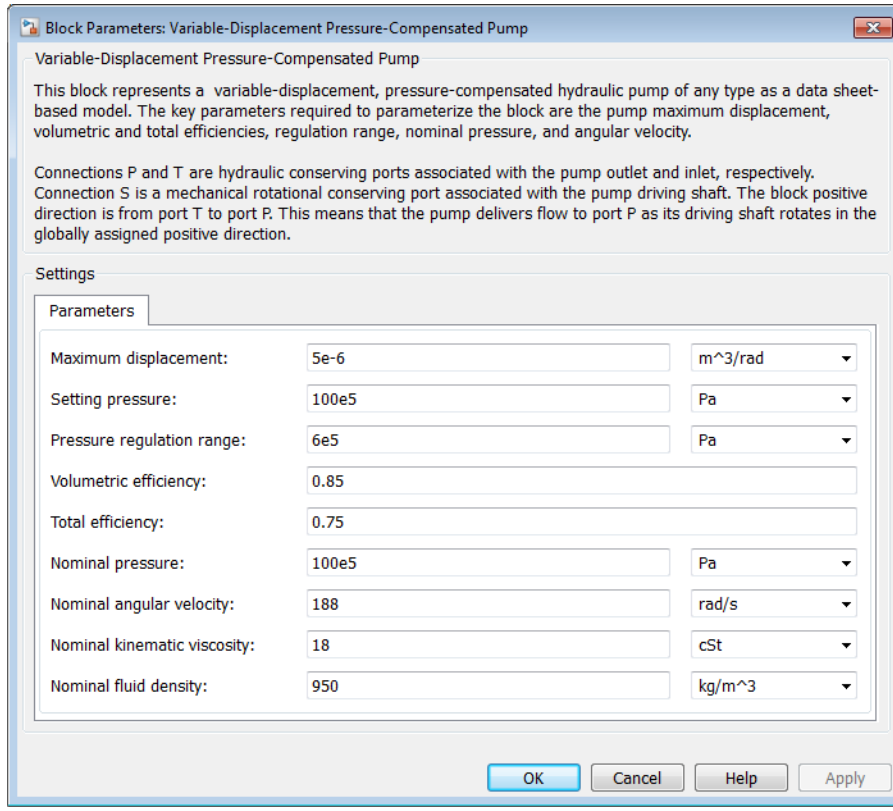
$$\eta_{mech} = \eta_{total} / \eta_V$$

The block positive direction is from port T to port P. This means that the pump transfers fluid from T to P provided that the shaft S rotates in the positive direction. The pressure differential across the pump is determined as $p = p_P - p_T$.

Basic Assumptions and Limitations

- Fluid compressibility is neglected.
- No loading on the pump shaft, such as inertia, friction, spring, and so on, is considered.
- Leakage inside the pump is assumed to be linearly proportional to its pressure differential.

Dialog Box and Parameters



Maximum displacement

Pump displacement. The default value is `5e-6` m^3/rad.

Setting pressure

Pump pressure setting. The default value is `1e7` Pa.

Pressure regulation range

Pressure range required to change the pump displacement from its maximum to zero. The default value is `6e5` Pa.

Volumetric efficiency

Pump volumetric efficiency specified at nominal pressure, angular velocity, and fluid viscosity. The default value is `0.85`.

Total efficiency

Pump total efficiency, which is determined as a ratio between the hydraulic power at the pump outlet and mechanical power at the driving shaft at nominal pressure, angular velocity, and fluid viscosity. The default value is `0.75`.

Nominal pressure

Pressure differential across the pump, at which both the volumetric and total efficiencies are specified. The default value is `1e7` Pa.

Nominal angular velocity

Angular velocity of the driving shaft, at which both the volumetric and total efficiencies are specified. The default value is `188` rad/s.

Nominal kinematic viscosity

Working fluid kinematic viscosity, at which both the volumetric and total efficiencies are specified. The default value is `18` cSt.

Nominal fluid density

Working fluid density, at which both the volumetric and total efficiencies are specified. The default value is `900` kg/m^3.

Global Parameters

Parameter determined by the type of working fluid:

- **Fluid kinematic viscosity**

Use the Hydraulic Fluid block or the Custom Hydraulic Fluid block to specify the fluid properties.

Ports

The block has the following ports:

`T`

Hydraulic conserving port associated with the pump suction, or inlet.

`P`

Hydraulic conserving port associated with the pump outlet.

S

Mechanical rotational conserving port associated with the pump driving shaft.

Examples

The Closed-Loop Actuator with Variable-Displacement Pressure-Compensated Pump example illustrates the use of the Variable-Displacement Pressure-Compensated Pump block in hydraulic systems.

See Also

Centrifugal Pump | Fixed-Displacement Pump | Variable-Displacement Pump

Introduced in R2006a

Variable-Displacement Pump

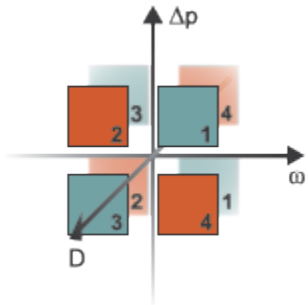
Variable-displacement bidirectional hydraulic pump

Library: Simscape / Fluids / Hydraulics (Isothermal) / Pumps and Motors

Description

The Variable-Displacement Pump block represents a device that extracts power from a mechanical rotational network and delivers it to a hydraulic (isothermal liquid) network. The pump displacement varies in proportion to the physical signal specified at port C or D. The exact port used depends on the block variant selected. See “Ports” on page 1-1373.

Ports **T** and **P** represent the pump inlets. Port **S** represents the pump drive shaft. During normal operation, the pressure gain from port **T** to port **P** is positive if the angular velocity at port **S** is positive also. This operation mode is referred to here as *forward pump*.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure gain from port **T** to port **P** (Δp), the angular velocity at port **S** (ω), and on the

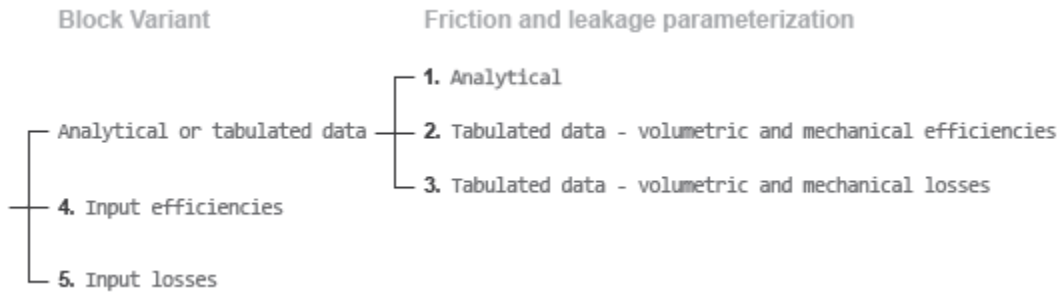
instantaneous displacement of the component (*D*). The “Operation Modes” on page 1-1364 figure maps the modes to the octants of a Δ*p*-*ω*-*D* chart. The modes are labeled 1–4:

- Mode **1**: forward pump — A positive shaft angular velocity generates a positive pressure gain.
- Mode **2**: reverse motor — A negative pressure drop (shown in the figure as a positive pressure gain) generates a negative shaft angular velocity.
- Mode **3**: reverse pump — A negative shaft angular velocity generates a negative pressure gain.
- Mode **4**: forward motor — A positive pressure drop (shown in the figure as a negative pressure gain) generates a positive shaft angular velocity.

The response time of the pump is considered negligible in comparison with the system response time. The pump is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The pump model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the pump inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Displacement Parameterizations

The displacement volume input depends on the block variant selected. If the active block variant is `Input efficiencies` or `Input losses`, the block obtains the instantaneous displacement volume directly from the physical signal input at port D.

If the active block variant is `Analytical or tabulated data`, the block computes the instantaneous displacement volume from the control member position specified at port C. This computation depends on the **Displacement parameterization** parameter setting:

- `Maximum displacement and control member stroke` — Compute the displacement volume per unit rotation as a linear function of the control member position specified at port C.
- `Displacement vs. control member position table` — Compute the displacement volume per unit volume using interpolation or extrapolation of displacement tabular data specified at discrete control member positions.

Flow Rate and Driving Torque

The volumetric flow rate generated at the pump is

$$q = q_{\text{Ideal}} + q_{\text{Leak}},$$

where:

- q is the net volumetric flow rate.
- q_{Ideal} is the ideal volumetric flow rate.
- q_{Leak} is the internal leakage volumetric flow rate.

The driving torque required to power the pump is

$$\tau = \tau_{\text{Ideal}} + \tau_{\text{Friction}},$$

where:

- τ is the net driving torque.
- τ_{Ideal} is the ideal driving torque.
- τ_{Friction} is the friction torque.

The ideal volumetric flow rate is

$$q_{\text{Ideal}} = D_{\text{Sat}} \cdot \omega,$$

and the ideal driving torque is

$$\tau_{\text{Ideal}} = D_{\text{Sat}} \cdot \Delta p,$$

where:

- D_{Sat} is a smoothed displacement computed so as to remove numerical discontinuities between negative and positive displacements.
- ω is the instantaneous angular velocity of the rotary shaft.
- Δp is the instantaneous pressure gain from inlet to outlet.

The saturation displacement depends on the block variant selected. If the active variant is `Analytical or tabulated data`,

$$D_{\text{Sat}} = \begin{cases} \text{sign}(D) \cdot D_{\text{Max}}, & |D| \geq D_{\text{Max}} \\ \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases},$$

where:

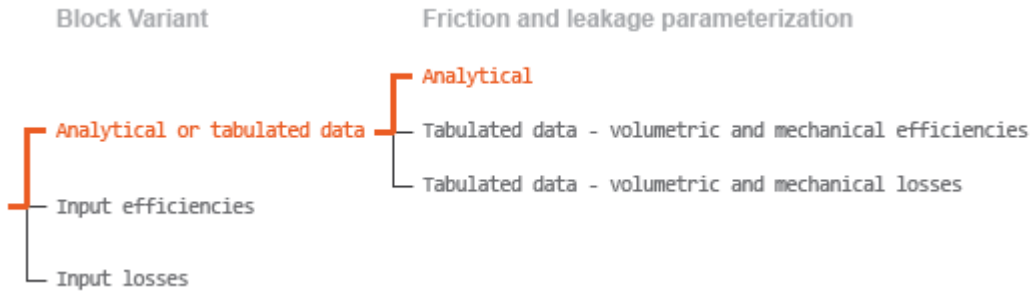
- D is the instantaneous fluid displacement determined from the physical signal input specified at port **C** or port **D**.

- D_{Max} is the specified value of the **Maximum displacement** block parameter.
- $D_{\text{Threshold}}$ is the specified value of the **Displacement threshold for pump-motor transition** block parameter.

If the active variant is `Input efficiencies` or `Input losses`, there is no upper bound on the displacement input and the saturation displacement reduces to:

$$D_{Sat} = \begin{cases} \sqrt{D^2 + D_{Thresh}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{Thresh}^2}, & D < 0 \end{cases}.$$

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$q_{\text{Leak}} = K_{\text{HP}}\Delta p,$$

and the friction torque is

$$\tau_{\text{Friction}} = \left(\tau_0 + K_{\text{TP}} \left| \frac{D_{\text{Sat}}}{D_{\text{Max}}} \right| |\Delta p| \right) \tanh\left(\frac{4\omega}{\omega_{\text{Thresh}}} \right),$$

where:

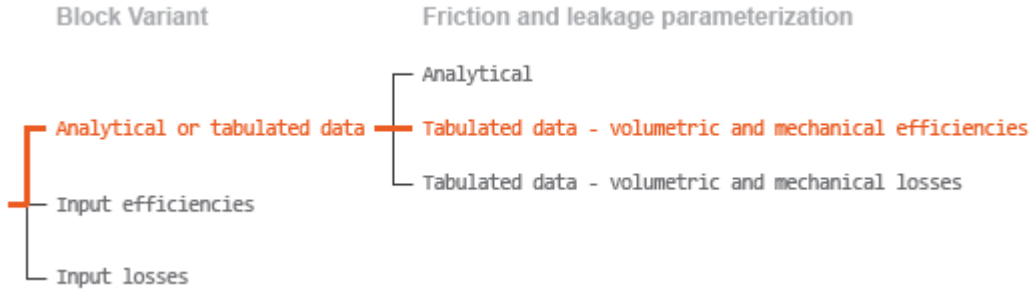
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- K_{TP} is the specified value of the **Friction torque vs pressure gain coefficient** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- $\omega_{\text{Threshold}}$ is the threshold angular velocity for the motor-pump transition. The threshold angular velocity is an internally set fraction of the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{\text{HP}} = \frac{\nu_{\text{Nom}}}{\rho \upsilon} \frac{\rho_{\text{Nom}} \omega_{\text{Nom}} D_{\text{Max}}}{\Delta p_{\text{Nom}}} \left(1 - \eta_{\text{v,Nom}} \right),$$

where:

- ν_{Nom} is the specified value of the **Nominal kinematic viscosity** block parameter. This is the kinematic viscosity at which the nominal volumetric efficiency is specified.
- ρ_{Nom} is the specified value of the **Nominal fluid density** block parameter. This is the density at which the nominal volumetric efficiency is specified.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- ρ is the actual fluid density in the attached hydraulic (isothermal liquid) network. This density can differ from the specified value of the **Nominal fluid density** block parameter.
- ν is the kinematic viscosity of the fluid associated with the fluid network.
- Δp_{Nom} is the specified value of the **Nominal pressure gain** block parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{\text{v,Nom}}$ is the specified value of the **Volumetric efficiency at nominal conditions** block parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`, the leakage flow rate is

$$q_{\text{Leak}} = q_{\text{Leak,Pump}} \frac{(1+\alpha)}{2} + q_{\text{Leak,Motor}} \frac{(1-\alpha)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Pump}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Motor}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the motor-pump transition.
- $q_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $q_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure drop threshold for motor-pump transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.
- $D_{\text{Threshold}}$ is the specified value of the **Displacement threshold for motor-pump transition** block parameter.

The leakage flow rate is computed from efficiency tabulated data through the equation

$$q_{\text{Leak,Pump}} = \left(1-\eta_{\text{v}}\right) q_{\text{Ideal}},$$

in pump mode and through the equation

$$q_{\text{Leak,Motor}} = -\left(1-\eta_{\text{v}}\right) q,$$

in motor mode, where:

- η_{v} is the volumetric efficiency obtained through interpolation or extrapolation of the **Volumetric efficiency table, e_v(dp,w,D)** parameter data.

Similarly, the friction torque is computed from efficiency tabulated data through the equation

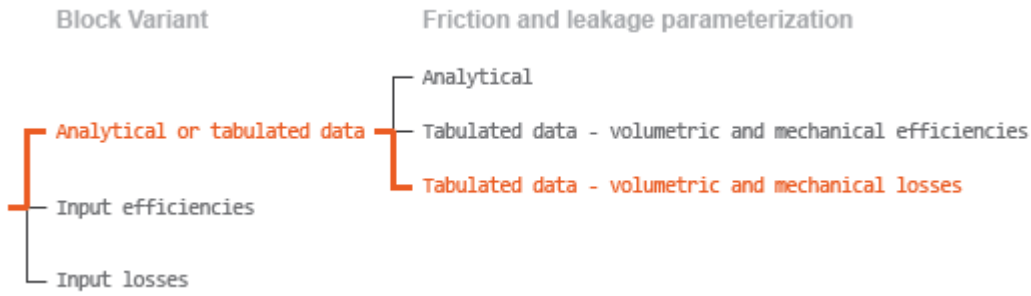
$$\tau_{\text{Friction,Pump}} = \left(1-\eta_{\text{m}}\right) \tau,$$

in pump mode and through the equation

$$\tau_{\text{Friction,Motor}} = -\left(1-\eta_{\text{m}}\right) \tau_{\text{Ideal}},$$

in motor mode, where:

- η_{m} is the mechanical efficiency obtained through interpolation or extrapolation of the **Mechanical efficiency table, e_m(dp,w,D)** parameter data.



Case 3: Loss Tabulated Data

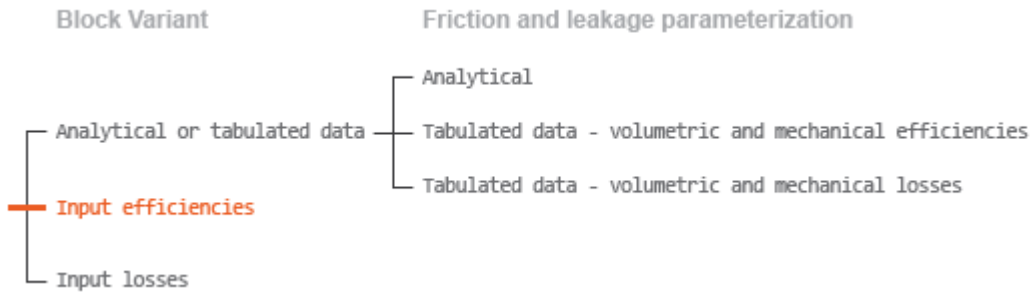
If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical losses`, the leakage flow rate equation is

$$q_{\text{Leak}} = q_{\text{Leak}}\left(\Delta p, \omega, D_{\text{Sat}}\right).$$

and the friction torque equation is

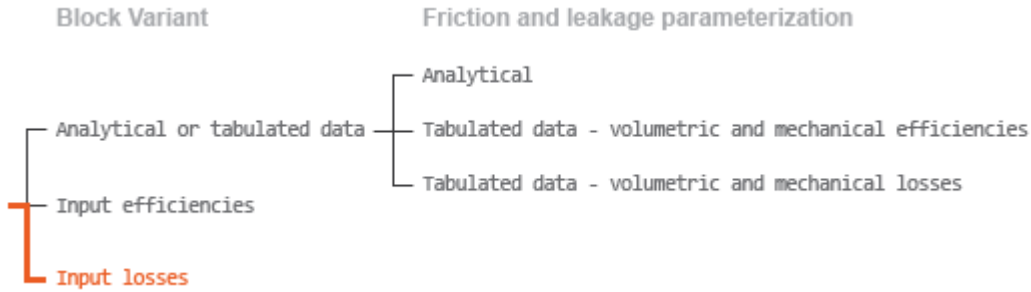
$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega, D_{\text{Sat}}\right),$$

where $q_{\text{Leak}}(\Delta p,\omega,D_{\text{Sat}})$ and $\tau_{\text{Friction}}(\Delta p,\omega,D_{\text{Sat}})$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the **Volumetric loss table, q_loss(dp,w,D)** and **Mechanical loss table, torque_loss (dp,w,D)** parameter data.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

Assumptions

- Fluid compressibility is negligible.
- Loading on the pump shaft due to inertia, friction, and spring forces is negligible.

Ports

Input

`C` — Control member offset, `m`

physical signal

Physical signal input port for the control member position. The block maps the control member offset to the corresponding displacement volume using the tabulated data specified in the block dialog box.

Dependencies

This port is exposed only when the block variant is set to `Analytical or tabulated data`.

`D` — Displacement volume, `cm^3/rev`

physical signal

Physical signal input port for the volume of fluid displaced per unit rotation. A smoothing function eases the transition between positive and negative input values.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies` or `Input losses`.

`EV` — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

`EM` — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

`LV` — Volumetric loss, `m^3/s`

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the pump inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

`LM` — Mechanical loss, `N*m`
physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotating pump shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

`T` — Pump inlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the pump inlet.

`P` — Pump outlet
hydraulic (isothermal liquid)

Hydraulic (isothermal liquid) conserving port representing the pump outlet.

`S` — Pump shaft
mechanical rotational

Mechanical rotational conserving port representing the pump shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-1365.

Variant 1: Analytical or tabulated data

`Displacement parameterization` — Parameterization used to map control positions to displacements
`Maximum displacement and control member stroke` (default)

Parameterization used to convert the physical signal at input port C to an instantaneous displacement:

- `Maximum displacement and control member stroke` — Compute the displacement as a linear function of the control position specified at port C.
- `Displacement vs. control member position table` — Obtain the displacement through interpolation or extrapolation of tabulated data specified over a range of control positions.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data`.

Restricted Parameter

This parameter is locked for editing when using the Simscape Restricted mode.

`Maximum stroke` — Control position corresponding to maximum displacement

`0.005 m` (default) | scalar with units of length

Control position corresponding to a maximum displacement. The physical signal input at port C saturates at this value. If the input rises higher, the block sets the control position to the maximum stroke.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data`.

`Maximum displacement` — Maximum fluid volume swept per unit shaft rotation

`30 cm^3/rev` (default) | scalar with units of volume/angle

Maximum fluid volume swept per unit shaft rotation. The displacement reaches this value when the control position hits the maximum stroke.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Maximum displacement and control member stroke`.

`Control member position vector` — Control positions at which to specify displacement tabular data
`[-0.0075, -0.0025, 0.0, 0.0025, 0.0075] m` (default) | *M*-element vector with units of length

M-element vector of control positions at which to specify the instantaneous displacement. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

`Displacement vector` — Displacement tabular data for the specified control positions
`[-30.0, -19.0, 0.0, 19.0, 30.0] cm^3/rev` (default) | *M*-element vector with units of volume/angle

M-element vector with the displacement tabular data for the specified control positions. The vector size, *M*, must match that of the **Control member position vector** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

`Interpolation method` — Numerical technique used to determine displacements within the tabular data range
`Linear` (default) | `Smooth`

Numerical technique used to map control position signals to displacements within the tabular date range. The interpolation method joins tabular data points using straight or curved line segments. Displacements within the range of the tabular data are assumed to lie on these segments.

- `Linear` — Join neighboring data points using straight line segments. Line slopes are generally discontinuous at the line segment end points.
- `Smooth` — Join neighboring data points using curved line segments generated with a modified Akima algorithm. Line slopes are continuous at the line segment end points.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

Restricted Parameter

This parameter is locked for editing when using the Simscape Restricted mode.

`Extrapolation method` — Numerical technique used to determine displacements outside the tabular data range

`Linear` (default) | `Nearest`

Numerical technique used to map control position signals to displacements outside the tabular date range. The extrapolation method extends the first and last tabular data points outward using horizontal or sloped straight line segments. Displacements outside the range of the tabular data are assumed to lie on these segments.

- `Linear` — Extend the tabular data using sloped straight line segments. The line slopes are computed from the first and last two tabular data points.
- `Nearest` — Extend the tabular data using horizontal straight line segments. The lines correspond to the displacements specified in the first and last tabular data points.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Displacement parameterization** parameter is set to `Displacement vs. control member position table`.

Restricted Parameter

This parameter is locked for editing when using the Simscape Restricted mode.

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque

`Analytical` (default)

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup tables to map pressure drop, angular velocity, and displacement to component efficiencies or losses. The tabular options include:

- `Tabulated data – volumetric and mechanical efficiencies`
- `Tabulated data – volumetric and mechanical losses`

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data`.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the volumetric efficiency

`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotating shaft at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure gain` — Pressure gain at which to specify the volumetric efficiency

`100e5 Pa` (default)

Pressure gain from inlet to outlet at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal kinematic viscosity` — Kinematic viscosity at which to specify the volumetric efficiency

`18 cSt` (default) | scalar with units of area/time

Kinematic viscosity of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating

conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal fluid density` — Fluid density at which to specify the volumetric efficiency

`900 kg/m^3` (default) | scalar with units of mass/volume

Mass density of the hydraulic fluid at which the component's nominal volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions

`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation

`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure gain coefficient at maximum displacement` — Proportionality constant at maximum displacement between friction torque and pressure gain
`0.6e-6 N*m/Pa` (default) | scalar with units of torque/pressure

Proportionality constant at maximum displacement between the friction torque on the mechanical shaft and the pressure gain from inlet to outlet.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Displacement threshold for pump–motor transition` — Displacement at which to initiate the transition between pumping and motoring modes
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

`Pressure gain vector for efficiencies, dp` — Pressure gains at which to specify the volumetric and mechanical efficiencies

M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies

N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Displacement vector for efficiencies, D` — Displacements at which to specify the volumetric and mechanical efficiencies

L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

`Volumetric efficiency table, e_v(dp,w,D)` — Volumetric efficiencies at the specified pressure gains, angular velocities, and displacements
unitless *M*-by-*N*-by*L* matrix

M-by-*N*-by-*L* matrix with the volumetric efficiencies at the specified fluid pressure gains, shaft angular velocities, and displacements. The efficiencies must fall in the range of 0–1. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w,D)` — Mechanical efficiencies at the specified pressure drops, angular velocities, and displacements
unitless *M*-by-*N* matrix

M-by-*N*-by-*L* matrix with the mechanical efficiencies corresponding to the specified fluid pressure gains, shaft angular velocities, and displacements. The efficiencies must fall in the range of 0–1. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Pressure gain threshold for pump–motor transition` — Pressure gain at which to initiate a smooth transition between motor and pump modes
`1e5 Pa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Angular velocity threshold for pump–motor transition` — Shaft angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies` or when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Check if operating beyond the octants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure gain, shaft angular velocity, or instantaneous displacement cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated`

`data — volumetric and mechanical efficiencies` or `Tabulated data — volumetric and mechanical losses`.

`Pressure gain vector for losses, dp` — Pressure gains at which to specify the volumetric and mechanical losses
M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses
N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Displacement vector for losses, D` — Displacements at which to specify the volumetric and mechanical losses
L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

`Volumetric loss table, q_loss(dp,w,D)` — Internal leakage flow rates at the specified pressure gains, angular velocities, and displacements
M-by-*N*-by-*L* matrix with units of volume/time

M-by-*N*-by-*L* matrix with the volumetric gains at the specified fluid pressure drops, shaft angular velocities, and displacements. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Mechanical loss table, torque_loss(dp,w,D)` — Friction torques at the specified pressure gains, angular velocities, and displacements

M-by-*N*-by-*L* matrix with units of torque

M-by-*N*-by-*L* matrix with the mechanical losses at the specified fluid pressure gains, shaft angular velocities, and displacements. Mechanical loss is defined here as the friction torque due to seals and internal components. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

Dependencies

This parameter is enabled when the block variant is set to `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Pressure gain threshold for pump–motor transition` — Pressure gain at which to initiate a smooth transition between motor and pump modes
`1e5 Pa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for pump–motor transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Displacement threshold for motor-pump transition` — Displacement at which to initiate a smooth transition between pump and motor modes
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal
`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

Variant 3: Input losses

`Displacement threshold for pump–motor transition` — Displacement below which the transition between pumping and motoring modes begins
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Check if operating beyond the pump mode` — Simulation warning mode for operating conditions outside pumping mode
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the pumping mode. A warning is issued if the pump transitions to motoring mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the block variant is set to `Input losses`.

`Check if lower side pressure violating minimum valid condition` — Simulation warning mode for minimum valid pressure
`None` (default) | `Warning`

Simulation warning mode for invalid pressures at the component ports. Select `Warning` to be notified when pressure falls below a minimum specified value. The warning can be useful in models where pressure can fall below the saturated vapor pressure of the hydraulic fluid, causing cavitation to occur.

`Minimum valid pressure` — Pressure required to trigger simulation warning
`0` (default) | scalar with units of pressure

Lower bound of the pressure validity range. A warning is issued if pressure falls below the specified value.

Dependencies

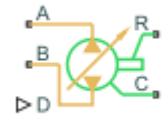
This parameter is enabled when the **Check if lower side pressure violating minimum valid condition** parameter is set to `Warning`.

See Also

Fixed-Displacement Motor | Fixed-Displacement Motor (TL) | Fixed-Displacement Pump | Fixed-Displacement Pump (TL) | Variable-Displacement Motor

Variable-Displacement Pump (TL)

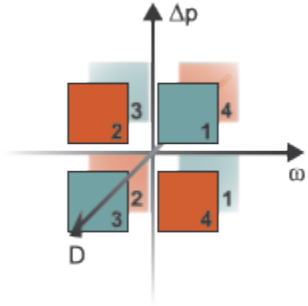
Variable-displacement bidirectional thermal liquid pump
Library: Simscape / Fluids / Thermal Liquid / Pumps & Motors



Description

The Variable-Displacement Pump block represents a device that extracts power from a mechanical rotational network and delivers it to a thermal liquid network. The pump displacement varies during simulation according to the physical signal input specified at port **D**.

Ports **A** and **B** represent the pump inlets. Ports **R** and **C** represent the motor drive shaft and case. During normal operation, the pressure gain from port **A** to port **B** is positive if the angular velocity at port **R** relative to port **C** is positive also. This operation mode is referred to here as *forward pump*.



Operation Modes

A total of four operation modes are possible. The working mode depends on the pressure gain from port A to port B (Δp), the angular velocity at port R relative to port C (ω), and

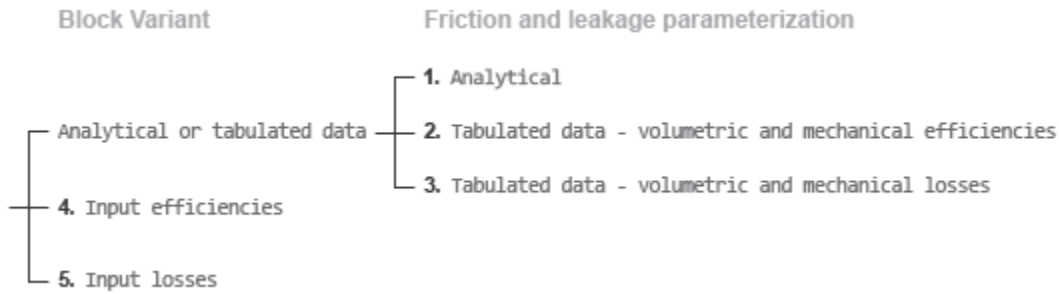
the instantaneous displacement input at port D (*D*). The “Operation Modes” on page 1-1364 figure maps the modes to the octants of a Δ*p*-*ω*-*D* chart. The modes are labeled 1–4:

- Mode **1**: forward pump — A positive shaft angular velocity generates a positive pressure gain.
- Mode **2**: reverse motor — A negative pressure drop (shown in the figure as a positive pressure gain) generates a negative shaft angular velocity.
- Mode **3**: reverse pump — A negative shaft angular velocity generates a negative pressure gain.
- Mode **4**: forward motor — A positive pressure drop (shown in the figure as a negative pressure gain) generates a positive shaft angular velocity.

The response time of the pump is considered negligible in comparison with the system response time. The pump is assumed to reach steady state nearly instantaneously and is treated as a quasi-steady component.

Block Variants and Loss Parameterizations

The pump model accounts for power losses due to leakage and friction. Leakage is internal and occurs between the pump inlet and outlet only. The block computes the leakage flow rate and friction torque using your choice of five loss parameterizations. You select a parameterization using block variants and, in the `Analytical or tabulated data` case, the **Friction and leakage parameterization** parameter.



Loss Parameterizations

The block provides three Simulink variants to select from. To change the active block variant, right-click the block and select **Simscape > Block choices**. The available variants are:

- `Analytical or tabulated data` — Obtain the mechanical and volumetric efficiencies or losses from analytical models based on nominal parameters or from tabulated data. Use the **Friction and leakage parameterization** parameter to select the exact input type.
- `Input efficiencies` — Provide the mechanical and volumetric efficiencies directly through physical signal input ports.
- `Input losses` — Provide the mechanical and volumetric losses directly through physical signal input ports. The mechanical loss is defined as the internal friction torque. The volumetric loss is defined as the internal leakage flow rate.

Flow Rate and Driving Torque

The mass flow rate generated at the pump is

$$\dot{m} = \dot{m}_{\mathrm{Ideal}} - \dot{m}_{\mathrm{Leak}},$$

where:

- $\dot{m}$ is the actual mass flow rate.
- $\dot{m}_{\mathrm{Ideal}}$ is the ideal mass flow rate.
- $\dot{m}_{\mathrm{Leak}}$ is the internal leakage mas flow rate.

The driving torque required to power the pump is

$$\tau = \tau_{\mathrm{Ideal}} + \tau_{\mathrm{Friction}},$$

where:

- τ is the actual driving torque.
- τ_{Ideal} is the ideal driving torque.
- τ_{Friction} is the friction torque.

The ideal mass flow rate is

$$\dot{m}_{\mathrm{Ideal}} = \rho D_{\mathrm{Sat}} \omega,$$

and the ideal generated torque is

$$\tau_{\text{Ideal}} = D_{\text{Sat}} \Delta p,$$

where:

- ρ is the average of the fluid densities at thermal liquid ports **A** and **B**.
- D_{Sat} is a smoothed displacement computed so as to remove numerical discontinuities between negative and positive displacements.
- ω is the shaft angular velocity.
- Δp is the pressure drop from inlet to outlet.

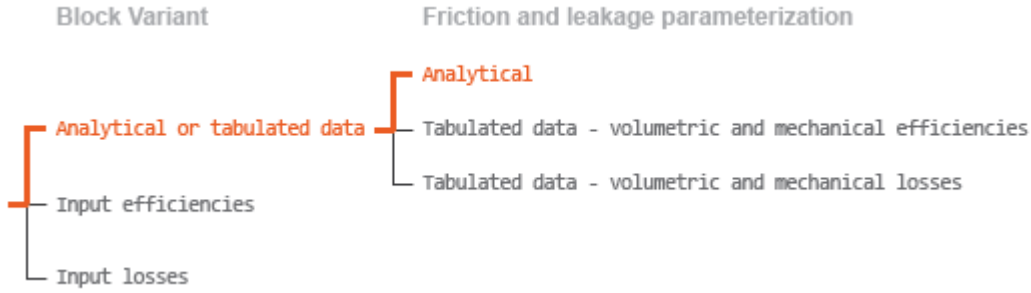
The saturation displacement is defined as:

$$D_{\text{Sat}} = \begin{cases} \sqrt{D^2 + D_{\text{Threshold}}^2}, & D \geq 0 \\ -\sqrt{D^2 + D_{\text{Threshold}}^2}, & D < 0 \end{cases}.$$

where:

- D is the displacement specified at physical signal port **D**.
- $D_{\text{Threshold}}$ is the specified value of the **Displacement threshold for motor-pump transition** block parameter.

The internal leakage flow rate and friction torque calculations depend on the block variant selected. If the block variant is `Analytical or tabulated data`, the calculations depend also on the **Leakage and friction parameterization** parameter setting. There are five possible permutations of block variant and parameterization settings.



Case 1: Analytical Efficiency Calculation

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Analytical`, the leakage flow rate is

$$\dot{m}_{\mathrm{Leak}} = \frac{K_{\mathrm{HP}}\rho_{\mathrm{Avg}}\Delta p}{\mu_{\mathrm{Avg}}},$$

and the friction torque is

$$\tau_{\mathrm{Friction}} = \left(\tau_0 + K_{\mathrm{TP}}\left|\Delta p\right|\frac{\left|D_{\mathrm{Sat}}\right|}{D_{\mathrm{Nom}}}\tanh\frac{4\omega}{(5e-5)\omega_{\mathrm{Nom}}}\right),$$

where:

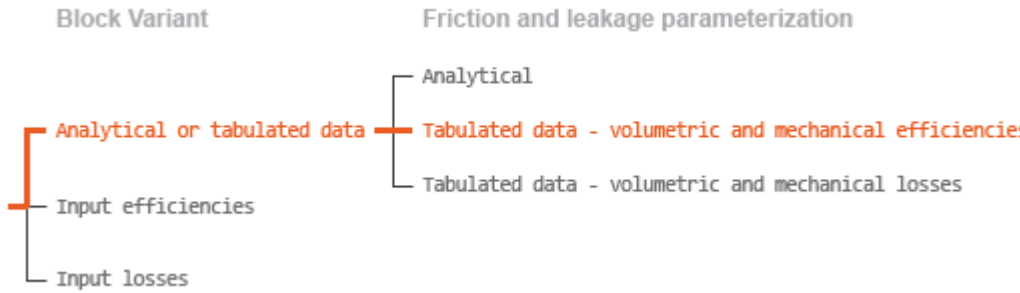
- K_{HP} is the Hagen-Poiseuille coefficient for laminar pipe flows. This coefficient is computed from the specified nominal parameters.
- μ is the dynamic viscosity of the thermal liquid, taken here as the average of its values at the thermal liquid ports.
- K_{TP} is the specified value of the **Friction torque vs pressure drop coefficient** block parameter.
- D_{Nom} is the specified value of the **Nominal Displacement** block parameter.
- τ_0 is the specified value of the **No-load torque** block parameter.
- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** block parameter.

The Hagen-Poiseuille coefficient is determined from nominal fluid and component parameters through the equation

$$K_{HP} = \frac{D_{Nom}\omega_{Nom}\mu_{Nom}\left(1-\eta_{v,Nom}\right)}{\Delta p_{Nom}},$$

where:

- ω_{Nom} is the specified value of the **Nominal shaft angular velocity** parameter. This is the angular velocity at which the nominal volumetric efficiency is specified.
- μ_{Nom} is the specified value of the **Nominal Dynamic viscosity** block parameter. This is the dynamic viscosity at which the nominal volumetric efficiency is specified.
- Δp_{Nom} is the specified value of the **Nominal pressure drop** block parameter. This is the pressure drop at which the nominal volumetric efficiency is specified.
- $\eta_{v,Nom}$ is the specified value of the **Volumetric efficiency at nominal conditions** block parameter. This is the volumetric efficiency corresponding to the specified nominal conditions.



Case 2: Efficiency Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`, the leakage flow rate is

$$\dot{m}_{Leak} = \dot{m}_{Leak,Pump}\frac{\left(1+\alpha\right)}{2} + \dot{m}_{Leak,Motor}\frac{\left(1-\alpha\right)}{2},$$

and the friction torque is

$$\tau_{\text{Friction}} = \tau_{\text{Friction,Pump}} \frac{1+\alpha}{2} + \tau_{\text{Friction,Motor}} \frac{1-\alpha}{2},$$

where:

- α is a numerical smoothing parameter for the motor-pump transition.
- $\dot{m}_{\text{Leak,Motor}}$ is the leakage flow rate in motor mode.
- $\dot{m}_{\text{Leak,Pump}}$ is the leakage flow rate in pump mode.
- $\tau_{\text{Friction,Motor}}$ is the friction torque in motor mode.
- $\tau_{\text{Friction,Pump}}$ is the friction torque in pump mode.

The smoothing parameter α is given by the hyperbolic function

$$\alpha = \tanh\left(\frac{4\Delta p}{\Delta p_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4\omega}{\omega_{\text{Threshold}}}\right) \cdot \tanh\left(\frac{4D}{D_{\text{Threshold}}}\right),$$

where:

- $\Delta p_{\text{Threshold}}$ is the specified value of the **Pressure gain threshold for pump-motor transition** block parameter.
- $\omega_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for pump-motor transition** block parameter.
- $D_{\text{Threshold}}$ is the specified value of the **Angular velocity threshold for motor-pump transition** block parameter.

The leakage flow rate is calculated from the volumetric efficiency, a quantity that is specified in tabulated form over the Δp–ω–D domain via the **Volumetric efficiency table** block parameter. When operating in pump mode (quadrants **1** and **3** of the Δp–ω–D chart shown in the “Operation Modes” on page 1-542 figure), the leakage flow rate is:

$$\dot{m}_{\text{Leak,Pump}} = \left(1-\eta_{\text{v}}\right)\dot{m}_{\text{Ideal}},$$

where η_{v} is the volumetric efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in motor mode (quadrants **2** and **4** of the Δp–ω–D chart), the leakage flow rate is:

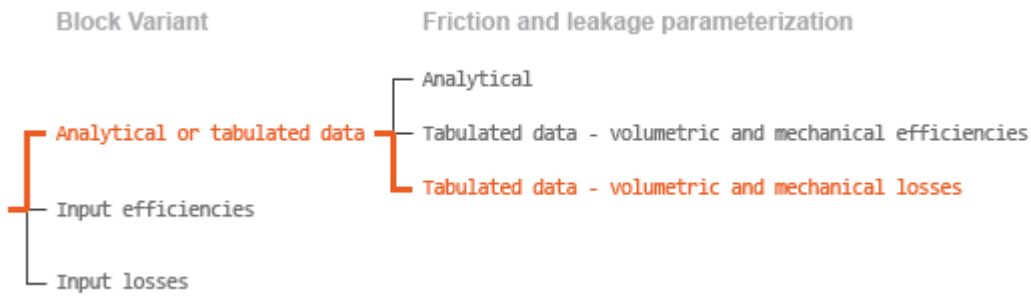
$$\dot{m}_{\text{Leak,Motor}} = -(1-\eta_{\text{v}})\dot{m}.$$

The friction torque is similarly calculated from the mechanical efficiency, a quantity that is specified in tabulated form over the Δp–ω–D domain via the **Mechanical efficiency table** block parameter. When operating in pump mode (quadrants **1** and **3** of the Δp–ω–D chart):

$$\tau_{\text{Friction,Pump}} = (1-\eta_{\text{m}})\tau,$$

where η_{m} is the mechanical efficiency, obtained either by interpolation or extrapolation of the tabulated data. Similarly, when operating in motor mode (quadrants **2** and **4** of the Δp–ω–D chart):

$$\tau_{\text{Friction,Motor}} = -(1-\eta_{\text{m}})\tau_{\text{Ideal}}.$$



Case 3: Loss Tabulated Data

If the active block variant is `Analytical or tabulated data` and the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`, the leakage (volumetric) flow rate is specified directly in tabulated form over the Δp–ω–D domain:

$$q_{\text{Leak}} = q_{\text{Leak}}(\Delta p, \omega, D_{\text{Sat}}).$$

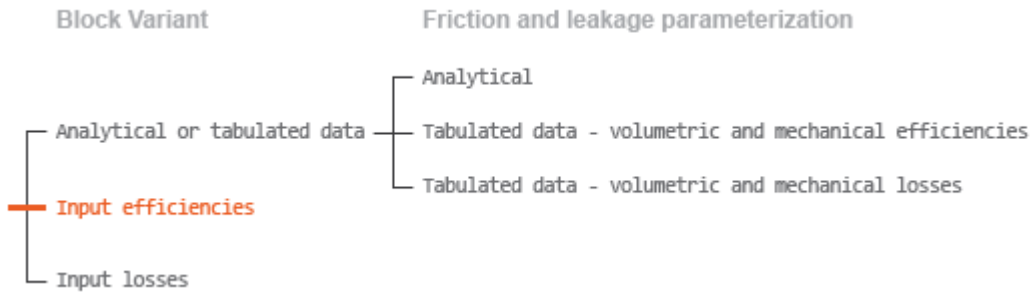
The mass flow rate due to leakage is calculated from the volumetric flow rate:

$$\dot{m}_{\text{Leak}} = \rho q_{\text{Leak}}.$$

The friction torque is similarly specified in tabulated form:

$$\tau_{\text{Friction}} = \tau_{\text{Friction}}\left(\Delta p, \omega, D_{\text{Sat}}\right),$$

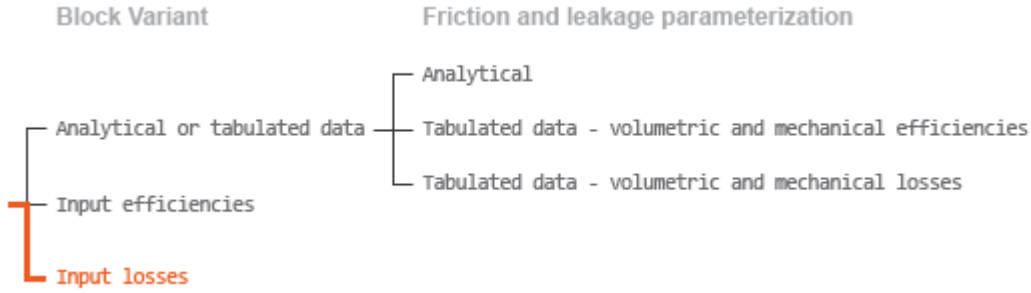
where $q_{\text{Leak}}(\Delta p,\omega)$ and $\tau_{\text{Friction}}(\Delta p,\omega)$ are the volumetric and mechanical losses, obtained through interpolation or extrapolation of the tabulated data specified via the **Volumetric loss table** and **Mechanical loss table** block parameters.



Case 4: Efficiency Physical Signal Inputs

If the active block variant is `Input efficiencies`, the leakage flow rate and friction torque calculations are as described for efficiency tabulated data (case 2). The volumetric and mechanical efficiency lookup tables are replaced with physical signal inputs that you specify through ports EV and EM.

The efficiencies are defined as positive quantities with value between zero and one. Input values outside of these bounds are set equal to the nearest bound (zero for inputs smaller than zero, one for inputs greater than one). In other words, the efficiency signals are *saturated* at zero and one.



Case 5: Loss Physical Signal Inputs

If the block variant is `Input losses`, the leakage flow rate and friction torque calculations are as described for loss tabulated data (case 3). The volumetric and mechanical loss lookup tables are replaced with physical signal inputs that you specify through ports LV and LM.

The signs of the inputs are ignored. The block sets the signs automatically from the operating conditions established during simulation—more precisely, from the Δp–ω quadrant in which the component happens to be operating. In other words, whether an input is positive or negative is irrelevant to the block.

Assumptions

- Fluid compressibility is negligible.
- Loading on the motor shaft due to inertia, friction, and spring forces is negligible.

Ports

Input

`D` — Displacement volume, `cm^3/rev`
physical signal

Physical signal input port for the volume of fluid displaced per unit rotation. A smoothing function eases the transition between positive and negative input values.

EV — Volumetric efficiency, unitless

physical signal

Physical signal input port for the volumetric efficiency coefficient. The input signal has an upper bound at the **Maximum volumetric efficiency** parameter value and a lower bound at the **Minimum volumetric efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

EM — Mechanical efficiency, unitless

physical signal

Physical signal input port for the mechanical efficiency coefficient. The input signal has an upper bound at the **Maximum mechanical efficiency** parameter value and a lower bound at the **Minimum mechanical efficiency** parameter value.

Dependencies

This port is exposed only when the block variant is set to `Input efficiencies`.

LV — Volumetric loss, `m^3/s`

physical signal

Physical signal input port for the volumetric loss, defined as the internal leakage flow rate between the pump inlets.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

LM — Mechanical loss, `N*m`

physical signal

Physical signal input port for the mechanical loss, defined as the friction torque on the rotary pump shaft.

Dependencies

This port is exposed only when the block variant is set to `Input losses`.

Conserving

`A` — Pump inlet
thermal liquid

Thermal liquid conserving port representing the pump inlet.

`B` — Pump outlet
thermal liquid

Thermal liquid conserving port representing the pump outlet.

`C` — Pump case
mechanical rotational

Mechanical rotational conserving port representing the pump case.

`R` — Pump shaft
mechanical rotational

Mechanical rotational conserving port representing the rotary pump shaft.

Parameters

The exposed block parameters depend on the active block variant. See “Block Variants and Loss Parameterizations” on page 1-1365.

Variant 1: Analytical or tabulated data

`Leakage and friction parameterization` — Parameterization used to compute leakage flow rate and friction torque
`Analytical` (default) | `Tabulated data — volumetric and mechanical efficiencies` | `Tabulated data — volumetric and mechanical losses`

Parameterization used to compute flow-rate and torque losses due to internal leaks and friction. The `Analytical` parameterization relies on nominal parameters generally available from component data sheets. The remaining, tabular, options rely on lookup tables to map pressure drop, angular velocity, and displacement to component efficiencies or losses.

`Nominal displacement` — Displacement at which to specify the volumetric efficiency

`30 cm^3/rev` (default) | scalar with units of volume/angle

Fluid displacement at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal shaft angular velocity` — Shaft angular velocity at which to specify the volumetric efficiency

`188 rad/s` (default) | scalar with units of angle/time

Angular velocity of the rotary shaft at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using simple linear functions, the leakage flow rate and friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal pressure gain` — Pressure gain at which to specify the volumetric efficiency

`10 MPa` (default)

Pressure drop from inlet to outlet at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Nominal dynamic viscosity` — Dynamic viscosity at which to specify the volumetric efficiency

`0.9 cP` (default) | scalar with units of area/time

Dynamic viscosity of the hydraulic fluid at which the component's volumetric efficiency is known. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Volumetric efficiency at nominal conditions` — Volumetric efficiency at the specified nominal conditions

`0.92` (default) | unitless scalar between `0` and `1`

Volumetric efficiency, defined as the ratio of actual to ideal volumetric flow rates, at the specified nominal conditions. Nominal parameters are typically published for standard operating conditions in manufacturer's data sheets. The block uses this parameter to calculate, using a simple linear function, the internal leakage flow rate.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`No-load torque` — Minimum torque required to induce shaft rotation

`0.05 N*m` (default) | scalar with units of torque

Torque required to overcome seal friction and induce rotation of the mechanical shaft. This torque is the load-independent component of the total friction torque.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Friction torque vs. pressure gain coefficient at nominal displacement` — Proportionality constant at maximum displacement between friction torque and pressure gain

`0.6 N*m/MPa` (default) | scalar with units of torque/pressure

Proportionality constant at maximum displacement between the friction torque on the mechanical shaft and the pressure gain from inlet to outlet.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Analytical`.

`Displacement threshold for pump-motor transition` — Fluid displacement at which to initiate a smooth transition between motor and pump modes

`0.5 cm^3/rev` (default) | positive scalar with units of volume/area

Fluid displacement below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet

`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Pressure gain vector for efficiencies, dp` — Pressure gains at which to specify the volumetric and mechanical efficiencies

M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the efficiency tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data – volumetric and mechanical efficiencies`.

`Shaft angular velocity vector for efficiencies, w` — Angular velocities at which to specify the volumetric and mechanical efficiencies

N-element vector with units of angular velocity (default)

N-element vector of shaft angular velocities at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Displacement vector for efficiencies, D` — Displacements at which to specify the volumetric and mechanical efficiencies

L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the efficiency tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Volumetric efficiency table, e_v(dp,w,D)` — Volumetric efficiencies at the specified pressure gains, angular velocities, and displacements

unitless *M*-by-*N*-by*L* matrix

M-by-*N*-by-*L* matrix with the volumetric efficiencies at the specified fluid pressure gains, shaft angular velocities, and displacements. The efficiencies must be in the range of `0`–`1`. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.

- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Mechanical efficiency table, e_m(dp,w,D)` — Mechanical efficiencies at the specified pressure gains, angular velocities, and displacements

unitless *M*-by-*N* matrix

M-by-*N*-by-*L* matrix with the mechanical efficiencies corresponding to the specified fluid pressure gains, shaft angular velocities, and displacements. The efficiencies must be in the range of `0`–`1`. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for efficiencies, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for efficiencies, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for efficiencies, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Pressure gain threshold for pump-motor transition` — Pressure gain at which to initiate a smooth transition between pump and motor modes

`1e-3 MPa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Angular velocity threshold for pump-motor transition` — Shaft angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Displacement threshold for pump-motor transition` — Fluid displacement at which to initiate a smooth transition between pump and motor modes
`0.5 cm^3/rev` (default) | positive scalar with units of volume/area

Fluid displacement below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies`.

`Check if operating beyond the octants of supplied tabulated data` — Simulation warning mode for operating conditions outside the range of tabulated data
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the range of tabulated data. Select `Warning` to be notified when the fluid pressure gain, shaft angular velocity, or instantaneous displacement cross outside the specified tabular data. The warning does not cause simulation to stop.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical efficiencies` or `Tabulated data — volumetric and mechanical losses`.

`Pressure gain vector for losses, dp` — Pressure gains at which to specify the volumetric and mechanical losses

M-element vector with units of pressure (default)

M-element vector of pressure gains at which to specify the loss tabular data. The vector size, *M*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Shaft angular velocity vector for losses, w` — Angular velocities at which to specify the volumetric and mechanical losses

N-element vector with units of angle/time (default)

N-element vector of shaft angular velocities at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must monotonically increase in value from left to right.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Displacement vector for losses, D` — Displacements at which to specify the volumetric and mechanical losses

L-element vector with units of volume/angle (default)

L-element vector of displacements at which to specify the loss tabular data. The vector size, *N*, must be two or greater. The vector elements need not be uniformly spaced. However, they must be monotonically increasing or decreasing.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants.

Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

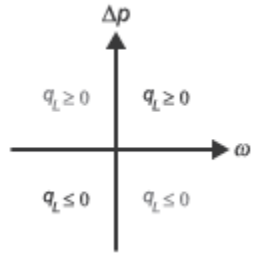
`Volumetric loss table, q_loss(dp,w,D)` — Internal leakage flow rates at the specified pressure gains, angular velocities, and displacements

M-by-*N*-by-*L* matrix with units of volume/time

M-by-*N*-by-*L* matrix with the volumetric losses at the specified fluid pressure gains, shaft angular velocities, and displacements. Volumetric loss is defined here as the internal leakage volumetric flow rate between port A and port B. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants. The tabulated data for the volumetric losses must obey the convention shown in the figure, with positive values at positive pressure gains and negative values at negative pressure gains.



Dependencies

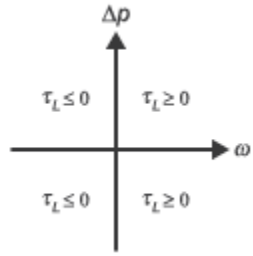
This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

`Mechanical loss table, torque_loss(dp,w,D)` — Friction torques at the specified pressure gains, angular velocities, and displacements
M-by-*N*-by-*L* matrix with units of torque

M-by-*N*-by-*L* matrix with the mechanical losses at the specified fluid pressure gains, shaft angular velocities, and displacements. Mechanical loss is defined here as the friction torque due to seals and internal components. *M*, *N*, and *L* are the sizes of the specified lookup-table vectors:

- *M* is the number of vector elements in the **Pressure gain vector for losses, dp** parameter.
- *N* is the number of vector elements in the **Shaft angular velocity vector for losses, w** parameter.
- *L* is the number of vector elements in the **Displacement vector for losses, D** parameter.

The tabulated data need not encompass all octants of operation—those of a (ω, Δp, D) chart. It suffices to specify the data for a single octant. Refer to the block description for the operation modes corresponding to the various octants. The tabulated data for the mechanical losses must obey the convention shown in the figure, with positive values at positive angular velocities and negative values at negative angular velocities.



Dependencies

This parameter is enabled when the **Leakage and friction parameterization** parameter is set to `Tabulated data — volumetric and mechanical losses`.

Variant 2: Input efficiencies

`Minimum volumetric efficiency` — Lower saturation bound on the volumetric efficiency input signal

`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal falls below the minimum volumetric efficiency, the volumetric efficiency is set to the minimum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum volumetric efficiency` — Upper saturation bound on the volumetric efficiency input signal

`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the volumetric efficiency. The input from physical signal port EV saturates at the specified value. If the input signal rises above the maximum volumetric efficiency, the volumetric efficiency is set to the maximum volumetric efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Minimum mechanical efficiency` — Lower saturation bound on the volumetric efficiency input signal
`1e-3` (default) | unitless scalar between `0` and `1`

Smallest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at the specified value. If the input signal falls below the minimum mechanical efficiency, the mechanical efficiency is set to the minimum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Maximum mechanical efficiency` — Upper saturation bound on the volumetric efficiency input signal
`1` (default) | unitless scalar between `0` and `1`

Largest allowed value of the mechanical efficiency. The input from physical signal port EM saturates at this value. If the input signal rises above the maximum mechanical efficiency, the mechanical efficiency is set to the maximum mechanical efficiency.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Pressure gain threshold for pump-motor transition` — Pressure gain at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Angular velocity threshold for motor-pump transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

Dependencies

This parameter is enabled when the block variant is set to `Input efficiencies`.

`Displacement threshold for pump-motor transition` — Displacement at which to initiate a smooth transition between pump and motor modes
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

Variant 3: Input losses

`Pressure gain threshold for pump-motor transition` — Pressure gain at which to initiate a smooth transition between motor and pump modes
`1e-3 MPa` (default) | scalar with units of pressure

Pressure gain from inlet to outlet below which the component begins to transition between pumping and motoring modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Angular velocity threshold for pump-motor transition` — Angular velocity at which to initiate a smooth transition between pump and motor modes
`10 rad/s` (default) | scalar with units of angle/time

Shaft angular velocity below which the component begins to transition between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Displacement threshold for pump-motor transition` — Displacement at which to initiate a smooth transition between pump and motor modes
`0.5 cm^3/rev` (default) | scalar with units of volume/angle

Absolute value of the instantaneous displacement below which the component transitions between motoring and pumping modes. A hyperbolic `Tanh` function transforms the leakage flow rate and friction torque such that the transition is continuous and smooth.

`Cross-sectional area at ports A and B` — Flow area at the component inlet and outlet
`0.01 m^2` (default) | positive scalar with units of area

Flow area at the component inlet and outlet. The areas are assumed equal. This parameter must be greater than zero.

`Check if operating beyond the motor mode` — Simulation warning mode for operating conditions outside motoring mode
`None` (default) | `Warning`

Simulation warning mode for operating conditions outside the motoring mode. A warning is issued if the motor transitions to pumping mode. Select `Warning` to be notified when this transition occurs. The warning does not cause simulation to stop.

Variables

`Mass flow rate into port A` — Mass flow rate into the component through thermal liquid port A
`1 kg/s` (default) | scalar with units of mass/time

Mass of fluid entering the component through the inlet per unit time at the start of simulation.

See Also

Centrifugal Pump (TL) | Fixed-Displacement Motor (TL) | Variable-Displacement Pump (TL)

Introduced in R2017b

Bibliography

[1] Cornell, C.R. “Dynamic Simulation of a Hydrostatically Propelled Vehicle.” SAE paper 811253, 1981, p. 22.

[2] “Flow of Fluids Through Valves, Fittings, and Pipe.” Crane Valves North America. Technical Paper No. 410.

[3] Hicks, T. G. and T. W. Edwards. *Pump Application Engineering*, New York: McGraw-Hill, 1971.

[4] Idelchik, I. E. *Handbook of Hydraulic Resistance*. New York: CRC Begell House, 1994.

[5] Karassic, I. J., J. P. Messina, P. Cooper, and C. C. Heald. *Pump Handbook*. Fourth edition. New York: McGraw-Hill, 2008.

[6] Keller, George R. *Hydraulic System Analysis*. Seattle, WA: Hydraulics & Pneumatics Magazine, 1970.

[7] Manring, Noah D. *Hydraulic Control Systems*. Hoboken, NJ: John Wiley & Sons, 2005.

[8] White, F. M. *Viscous Fluid Flow*. New York: McGraw-Hill, 1991.

[9] Yeapple, F. *Fluid Power Design Handbook*, New York: Marcel Dekker, 1995.

across variables — Variables that are measured with a gauge connected in parallel to an element.

add-on products — Products in the Physical Modeling family that use Simscape platform and, as a result, share common functionality such as physical units management, editing modes, and so on. Simscape Fluids software is one of the Simscape add-on products.

behavioral block implementation model — A block that is implemented based on its physical behavior, described by a system of mathematical equations. An example of a behavioral block implementation is the Variable Orifice block.

conserving ports — Bidirectional hydraulic or mechanical ports that represent physical connections and relate physical variables based on the Physical Network approach.

constructional block implementation model — A block that is constructed out of other blocks, connected in a certain way. An example of a constructional block implementation is the 4-Way Directional Valve block, which is constructed based on four Variable Orifice blocks.

data-sheet-based model — A block with a set of parameters determined by data that is usually listed in the manufacturer's catalogs or data sheets.

globally assigned positive direction — Direction considered positive for a model diagram.

nonrestricted parameters — Parameters that are available for modification when you open a model in Restricted mode. Usually, these are the block parameters with plain numerical values, such as **Pipe internal diameter** or **Resistance area**. Information on restricted and nonrestricted parameters is listed in block reference pages.

physical signal ports — Unidirectional ports (inports and outports) transferring signals that use an internal physical modeling engine for computations.

restricted parameters	Parameters that are not available for modification when you open a model in Restricted mode. You have to be in Full mode to modify them. Usually, these are the block parameterization options, such as **Pipe cross section type** or **Interpolation method**. Information on restricted and nonrestricted parameters is listed in block reference pages.
through variables	Variables that are measured with a gauge connected in series to an element.